



**SVHEAT<sup>TM</sup>**

**2D / 3D Geothermal Modeling Software**

# **Tutorial Manual**

**Written by:**

**Robert Thode, B.Sc.G.E., P.Eng.**

**Edited by:**

**Murray Fredlund, Ph.D., P.Eng.**

**SoilVision Systems Ltd.  
Saskatoon, Saskatchewan, Canada**

## **Software License**

The software described in this manual is furnished under a license agreement. The software may be used or copied only in accordance with the terms of the agreement.

## **Software Support**

Support for the software is furnished under the terms of a support agreement.

## **Copyright**

Information contained within this Tutorial Manual is copyrighted and all rights are reserved by SoilVision Systems Ltd. The SVHEAT software is a proprietary product and trade secret of SoilVision Systems. The Tutorial Manual may be reproduced or copied in whole or in part by the software licensee for use with running the software. The Tutorial Manual may not be reproduced or copied in any form or by any means for the purpose of selling the copies.

## **Disclaimer of Warranty**

SoilVision Systems Ltd. reserves the right to make periodic modifications of this product without obligation to notify any person of such revision. SoilVision does not guarantee, warrant, or make any representation regarding the use of, or the results of, the programs in terms of correctness, accuracy, reliability, currentness, or otherwise; the user is expected to make the final evaluation in the context of his (her) own problems.

## **Trademarks**

Windows™ is a registered trademark of Microsoft Corporation.  
SoilVision® is a registered trademark of SoilVision Systems Ltd.  
SVOFFICE™ is a trademark of SoilVision Systems Ltd.  
SVFLUX™ is a trademark of SoilVision Systems Ltd.  
CHEMFLUX™ is a trademark of SoilVision Systems Ltd.  
SVAIRFLOW™ is a trademark of SoilVision Systems Ltd.  
SVSOLID™ is a trademark of SoilVision Systems Ltd.  
SVHEAT™ is a trademark of SoilVision Systems Ltd.  
SVSLOPE® is a registered trademark of SoilVision Systems Ltd.  
ACUMESH™ is a trademark of SoilVision Systems Ltd.  
FlexPDE® is a registered trademark of PDE Solutions Inc.

Copyright © 2013  
by  
SoilVision Systems Ltd.  
Saskatoon, Saskatchewan, Canada  
ALL RIGHTS RESERVED  
Printed in Canada  
Last Updated: May 20, 2013

---

<b>1 Introduction</b>	<b>4</b>
<b>2 Authorization</b>	<b>5</b>
<b>3 1D Road pavement</b>	<b>6</b>
3.1 Model Setup	7
3.2 Results and Discussion.....	22
<b>4 1D Freezing Front Analysis.....</b>	<b>25</b>
4.1 Model Setup	26
4.2 Results and Discussion.....	30
4.3 Model Data	31
<b>5 2D Partition Model</b>	<b>33</b>
5.1 Model Setup	33
5.2 Results and Discussion.....	37
<b>6 2D Chilled Pipe</b>	<b>40</b>
6.1 Model Setup	41
6.2 Results and Discussion.....	45
<b>7 2D Heated Pipeline</b>	<b>50</b>
7.1 Model Setup	51
7.2 Results and Discussion.....	58
<b>8 2D Hairpin Thermosyphon.....</b>	<b>60</b>
8.1 Model Setup	61
8.2 Results and Discussion.....	70
8.3 Model Data	73
<b>9 2D Canal Bank Freezing and Thawing.....</b>	<b>75</b>
9.1 Model Setup	75
9.2 Results and Discussion.....	87
<b>10 3D Heated Foundation.....</b>	<b>95</b>
10.1 Model Setup	96
10.2 Results and Discussion.....	103
<b>11 References</b>	<b>105</b>

# 1 Introduction

The Tutorial Manual serves a special role in guiding the first time users of the SVHEAT software through a typical example problem. The example is "typical" in the sense that it is not too rigorous on one hand and not too simple on the other hand.

The Tutorial Manual serves as a guide by: assisting the user with the input of data necessary to solve the boundary value problem, ii.) explaining the relevance of the solution from an engineering standpoint, and iii.) assisting with the visualization of the computer output. An attempt has been made to ascertain and respond to questions most likely to be asked by first time users of SVHEAT.

## 2 Authorization

Certain features in SVOFFICE are only available in the FULL (PROFESSIONAL), version of the software. Perform the following steps to check if FULL authorization is activated:

1. Plug in the USB security key,
2. Select *File > Authorization...* from the menu on the SVOFFICE Project Manager,
3. The software should display full authorization under the *Level Authorized* heading. If not, the security codes provided by SoilVision Systems at the time of purchase have not yet been entered. Please see the [Authorization](#) section of the SVOFFICE User's Manual for instructions on entering these codes.

### 3 1D Road pavement

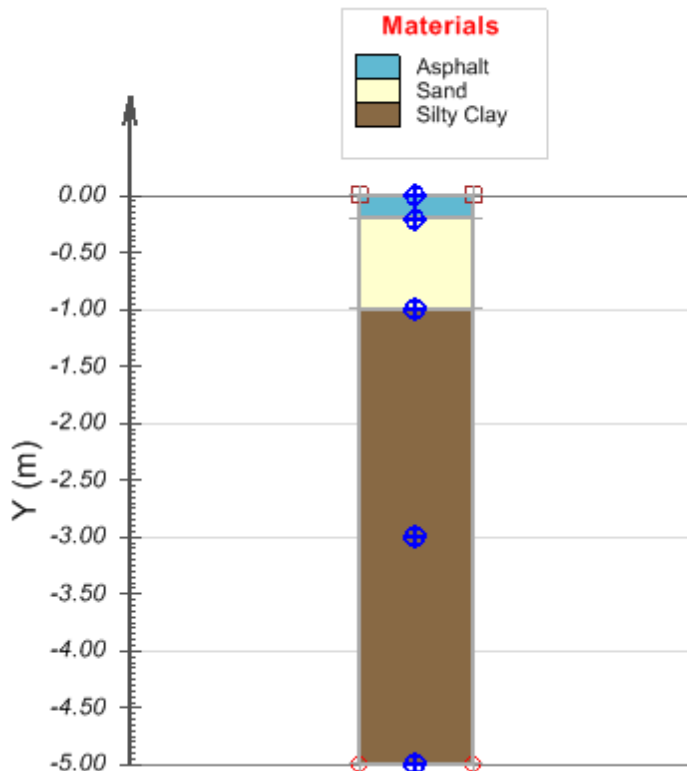
The purpose of this tutorial is to illustrate

- Procedures to create a 1D SVHeat model,
- Using different methods to specify thermal material properties,
- Applying the temperature as the climate boundary condition,
- Solve a transient problem of heat transfer.

Project: Roadways  
Model: RoadPavement\_TUT1D  
RoadPavement\_TUT1D\_Imperial  
RoadPavement\_TUT1D\_MinMaxTemp  
RoadPavement\_TUT1D\_HourlyTemp

Minimum authorization required: PROFESSIONAL

#### Model Description and Geometry



Region	Y coordinate
R1	0.0 to -0.2
R2	-0.2 to -1.0
R3	-1.0 to -5.0

## 3.1 Model Setup

The following steps are required in order to set up and solve the model described in the preceding section. The steps fall under the general categories of:

- a. Create model
- b. Create model geometry
- c. Specify model global settings
- d. Specify material properties
- e. Apply material to the each region
- f. Specify climate properties
- g. Apply boundary conditions
- h. Specify initial condition
- i. Specify model output
- j. Run model
- k. Visualize results
- l. Tune model settings

### NOTE:

Any values on the dialogs there are not specifically mentioned in the steps below are assumed to be the default values currently present.

### a. Create Model

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog.
2. Select the project called "UserTutorial". If the project does not exist, create a new project called "UserTutorial",
3. On the right side in "SVOOffice 2009 Manager" dialog, click "New" button. The "Create New Model" dialog is opened. In this dialog, enter data or select the settings as shown following:

In the **General** Tab the following settings must be specified.

Application:	Select " <b>SVHeat</b> " from the drop-down list,
Model Name:	Enter " <b>RoadPavement_TUT1D</b> " in the text box,
System:	Select " <b>1D Vertical</b> " from the drop-down list,
Type:	Select " <b>Transient</b> " from the drop-down list,
Units:	Select " <b>Metric</b> " from the drop-down list,
Time Units:	Select " <b>Day</b> " from the drop-down list.

In the **World Coordinate System** Tab

This tutorial is to simulate thermal transfer of soil below the road pavement. Enter Y coordinate as following:

	Minimum	Maximum
Y:	-5	0

In the **Time** Tab

The model is to simulate one year of climate temperature on the road pavement. Enter the end time of 365 days.

Start Time: 0  
 Initial Increment: 0.1  
 Maximum Increment: 0.2  
 End Time: 365

#### NOTE

:

If the maximum increment time step is set too large, it may effect FEM solution convergence.

Double check the above model settings. If no problem, then Click the **OK** button on the "Create New Model" dialog. After that, an "**Options**" dialog will be opened. Click "**OK**" button to close the dialog with the default settings.

### b. Create Model Geometry

Model geometry is defined as a set of regions. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

The model geometry is composed of 3 regions or layers, as shown in the following table:

Region Name	Y coordinate	Material
Region 1: R1	0 to -0.2 m	Asphalt
Region 2: R2	-0.2 to -1 m	Sand
Region 3: R3	-1 to -5 m	Silty Clay

A few methods can be used to create model geometry. One is to use the menu **Draw >> Model Geometry >> Region Polygon** to draw each region. Another is to use the 1D Thicknesses dialog (simplest). Another method is to enter the coordinates for each region. The coordinates for each region will be entered in this example.

The user must follow the steps below in order to enter the geometry correctly.

1. Click the menu of **Model >> Geometry >> Regions**. A dialog of "Regions" will be opened after clicking the menu.
2. Select the row with name of "**R1**" from the data grid table , and click "**Properties...**" button to edit the region properties.

#### NOTE

:

You can also open the **Regions Properties** dialog by double clicking the selected region in the data grid.

Region 1 (0 to -0.2 m)

3. In "**Region Properties**" dialog, click "**New Polygon...**" button,
4. Enter y coordinate of 0,  
After the value is entered, a new bank row will be added in the "New Polygon Shape" dialog,
5. Enter y coordinate -0.2 in the second row of data grid,
6. Click **OK** button,
7. Click **OK** button in "**Region Properties**" dialog.

Region 2 (-0.2 to -1 m)



8. Click **New** button in "**Regions**" dialog to create Region 2.
9. Select the row of **R2** from the data grid and then click the "**Properties...**" button.
10. In "**Region Properties**" dialog, click "**New Polygon...**" button
11. Enter y coordinate of -0.2
12. Enter y coordinate of -1 in the second row of data grid.
13. Click **OK** button
14. Click **OK** button in "**Region Properties**" dialog

Region 3 (-1 to -5 m)

15. Click **New** button in "**Regions**" dialog to create Region 3.
16. Select the row of **R3** from the data grid and then click the "**Properties...**" button.
17. In "**Region Properties**" dialog, click "**New Polygon...**" button
18. Enter y coordinate of -1
19. Enter y coordinate of -5
20. Click **OK** button
21. Click **OK** button in "**Region Properties**" dialog
22. Click **OK** button in "**Region**" dialog

### c. Specify Model Global Settings (Model >> Settings)

SVHeat global settings allows user to specify different features such as thermal conduction, thermal convection, ice phase change, etc. Furthermore, soil water content can be specified in different approaches. Click the menu of **Model >> Settings...** to open the dialog. However, this tutorial will use the default settings.

### d. Specify Material Properties (Model >> Material)

Three materials of asphalt, sand, and clay, will be used in the model. The following steps are to specify the thermal material for each material.

Click the menu of **Model >> Materials >> Manager.**

1. Click "**New**" button in the dialog of "**Materials Manager**",  
Enter the name for a material  
Name: "asphalt",
2. Click "**OK**" button in the "New Material" dialog.

Specify thermal Conductivity for for asphalt material

3. Click "**Conductivity**" tab to specify thermal conductivity,
4. Select "**Constant**" from the drop-down list of "**Thermal Conductivity Option**"  
Select the check box of "**Same value for unfrozen or frozen material**",  
Enter the constant value of thermal conductivity for asphalt  
**Unfrozen Material:** 103680 J/day-m-C,  
Thermal conductivity for asphalt material is estimated as 1.2 w/m-C, or 1.2 x 3600 x 24 = 103680 J/day-m-C.

Specify the Volumetric Heat Capacity for asphalt material

5. Click "**Volumetric Heat Capacity**" tab
6. Select "**Constant**" checkbox, and enter the value of heat capacity:  
Select the check box of "**Same value of unfrozen and frozen HC**",

Enter the value of heat capacity  
Frozen Volumetric Heat Capacity: 2520000 J/m<sup>3</sup>-C.

Specify Soil Freezing Characteristic curve (SFCC)

7. Click SFCC tab

Select "**None**" from the drop-down list of "**SFCC method**", because the phase change for asphalt material does not need to be considered.

Specify the Volumetric Water Content (VWC)

8. Click "**VWC**" tab,

The water content of asphalt can be neglected, Enter a small amount of water content,

SatVWC: 0.001

VWC: 0.001

9. Click **OK** button.

Specify Thermal properties for **sand** material

10. Click "**New**" button in the "Material Manager" dialog,

Enter material name: sand,

11. Click "**OK**" button in the "New Material" dialog

Specify Thermal Conductivity for **sand** material

12. Click "**Conductivity**" tab,

Select "**Johansen**" approach from the drop-down list,0

With the Johansen approach, the thermal conductivity is calculated based on water content, ice content and soil solid component fraction.

Material State: Crushed,

Material Type: Fine,

Solid Conductivity: checked,

Solid Component: 734400 J/day-m-C,

The estimated value of thermal conductivity solid Component = 734400 J/day-m-C.

Specify Volumetric Heat Capacity for sand material

13. Click "**Volumetric Heat Capacity**" tab,

Select the radio button of "**Jame-Newman**",

Solid Dry Density: 1600 kg/m<sup>3</sup>,

Specific Heat Capacity of Solid Component: 700 J/kg-C,

Specify Volumetric Water Content (VWC) of sand material

14. Click "**VWC**" tab,

Enter a constant value of sand water content,

SatVWC: 0.32

VWC: 0.2

Specify Soil Freezing Characteristic curve (SFCC)

15. Click SFCC tab

Enter the interval of phase change temperature for sand,

**From:** -0.01 C,  
**To:** -0.5 C

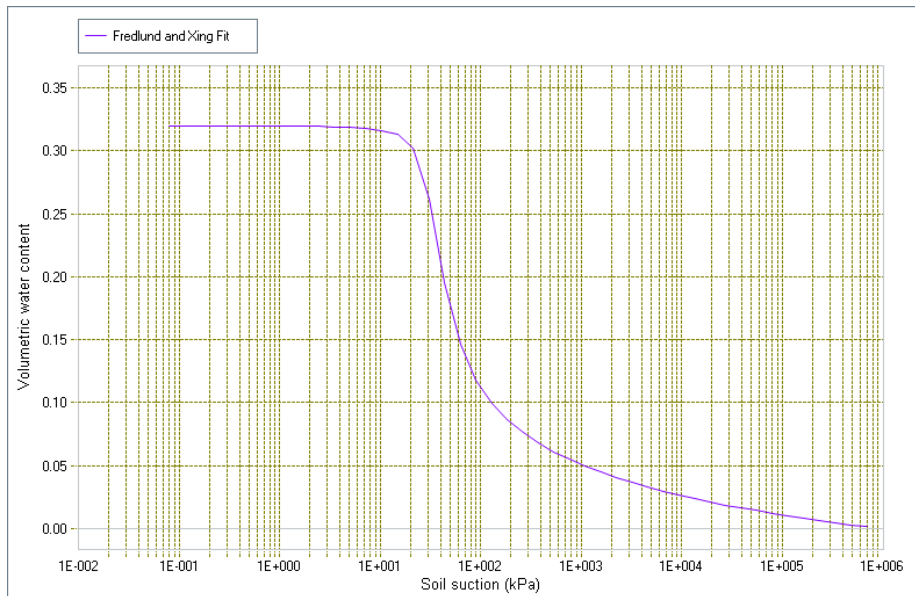
Select the "Estimated By SWCC" from drop-down list.

It is very important to specify the relationship between of unfrozen water content and the temperature. If data of unfrozen water content is unavailable, it can be estimated with soil water characteristic curve (SWCC).

Select *Fredlund and Xing Fit* from the drop-down list of SWCC Methods,  
Click the SWCC Properties... button,  
Enter the following value for the SWCC fitting parameter.

af: 29.4 kPa  
nf: 6.04  
mf: 0.335  
hr: 105.9 kPa

Click Graph SWCC... button to preview SWCC as shown in Figure 1.



**Figure SWCC that is used to estimate SFCC**

16. Click "OK" button to close the graph,
17. Click "OK" button to close the Material Properties dialog,

Specify thermal properties for **clay** material

18. Click "New" button in the "Material Manager" dialog,  
Enter material name: silty clay,
19. Click "OK" button in the "New Material" dialog

Specify Thermal Conductivity for for clay material

20. Select "DeVries" approach from the drop-down list,  
Enter the thermal Conductivity of each phase in soil.

Solid Phase: 708480 J/day-m-C  
Water Phase: 52272 J/day-m-C  
Dry air Phase: 2073.6 J/day-m-C  
Vapor air: 466560 kg/m<sup>3</sup>

Specify Volumetric Heat Capacity for clay material

21. Click "**Volumetric Heat Capacity**" tab,  
22. Select radio button of "**Jame-Newman**",  
Solid Dry Density: 1320 kg/m<sup>3</sup>  
Specific Heat Capacity of Solid Component: 800 J/kg-C

Specify Soil Freezing Characteristic curve (SFCC)

23. Click SFCC tab  
Enter the interval of phase change temperature for clay,  
**From:** -0.05 C,  
**To:** -1.50 C  
24. Select the "Tice & Anderson Fit" from "**SFCC Method**" drop-down list.  
Enter the value for parameter A: 0.035  
parameter B: 0.450  
Soil dry density: 1320 kg/m<sup>3</sup>

Specify Volumetric Water Content (VWC) of clay material

25. Click "**VWC**" tab,  
Enter a constant water content of clay material,  
SatVWC: 0.45 m<sup>3</sup>/m<sup>3</sup>  
**VWC:** 0.3 m<sup>3</sup>/m<sup>3</sup>  
26. Click "**OK**" button in the "**Material Properties**" dialog.  
27. Click "**OK**" button in the "**Material Manager**" dialog.

## e. Apply Material to Each Region

1. Click menu of **Model >> Geometry >> Regions**,
2. Select the **R1** region in the data grid,
3. Click "**Properties**" button,
4. Select "**asphalt**" from the "**Material:**" drop-down list on the bottom-left corner in the dialog,
5. Click "**OK**" button
6. Select "**R2**" region in the data grid,
7. Click "**Properties**" button,
8. Select "**sand**" from the "**Material:**" drop-down list on the bottom-left corner in the dialog,
9. Click "**OK**" button.

10. Select "**R3**" region in the data grid,
11. Click "**Properties**" button,
12. Select "**silty clay**" from the "**Material:**" drop-down list on the bottom-left corner in the dialog,
13. Click "**OK**" button in "**Region Properties**",
14. Click "**OK**" button in "**Regions**" dialog.

**NOT****E:**

You can open a particular region properties dialog by moving mouse onto a particular region, and then double clicking the region, as shown in Figure 2.

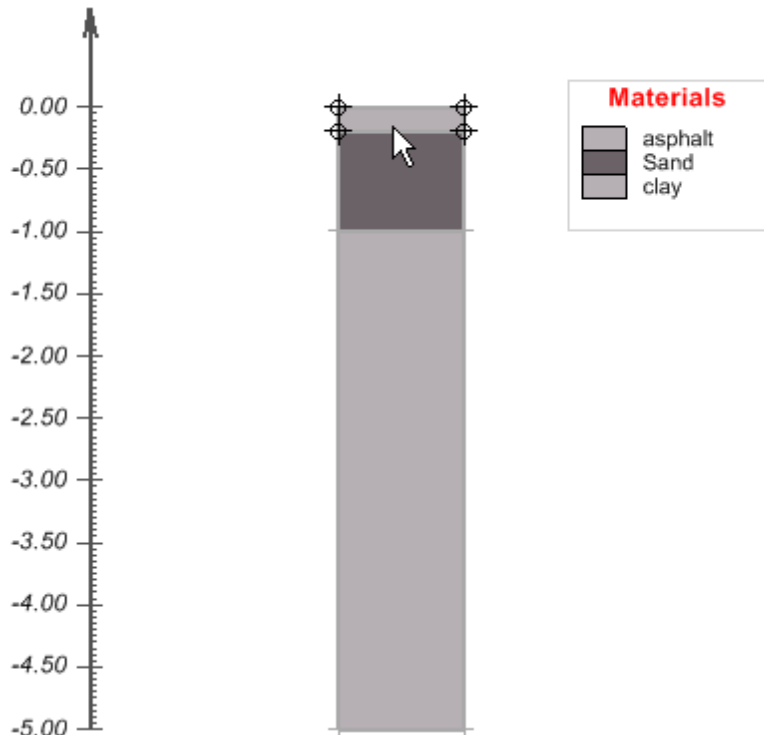


Figure 2 Quickly open a region properties dialog

## f. Specify Climate Properties (Model >> Boundaries >> Climate Manager)

To simulate climate temperature effects on heat transfer, use the climate manager to specify climate properties.

**NOT****E:**

Climate Manager requires the Professional License of SVOOffice2009. You can specify the air temperature by the Boundary Condition Settings dialog (see step g) for Standard License of SVOOffice2009.

1. Click the menu of Model >> Boundaries >> Climate Manager,

2. Click "New" button in the "Climate Manager" dialog,
3. Enter a climate name in the text box,  
Name: DemoClimate
4. Click "OK" button,

Specify an approach to determine the temperature at the ground surface

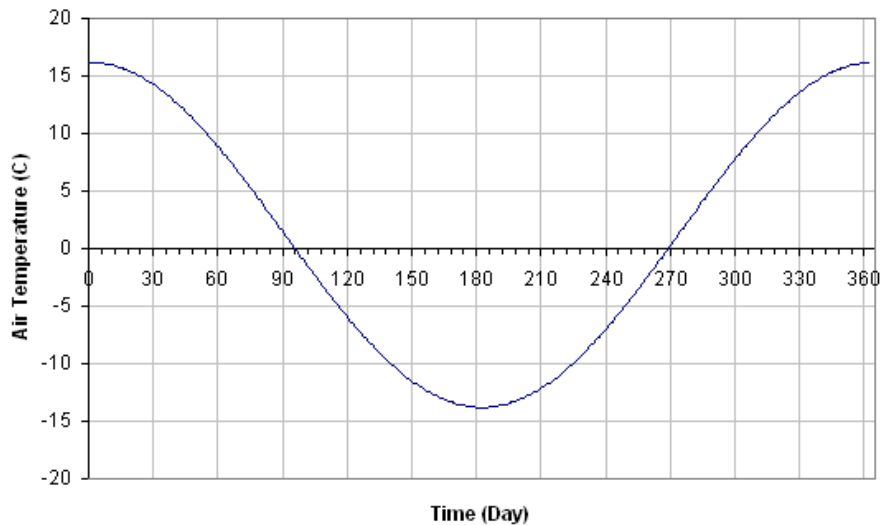
Generally, the temperature at the ground surface is different from the air temperature. In this tutorial, N-Factor approach is used.

5. Click "General" Tab
6. Select "Empirical with N-Factor" from the drop-down list,
7. Click "N-Factor" tab,
8. Select "Constant" from the drop-down list,
9. Enter an estimated value of N-Factor,  
N-Factor Constant: 0.8
10. Click "Air Temperature" Tab,
11. Select "Expression" from the drop-down list,
12. Enter the following formula to simulate daily changing temperature, shown in Figure 3.

Expression:  $1.2 + 15 * \sin(2 * 3.141596 / 365 * t + 3.141596 / 2)$

where:

$t$  = time. It is a valid variable in PDE script.



**Figure 3 Air temperature changing with time**

13. Click "OK" button in "Climate Properties" dialog,
14. Click "OK" button in "Climate Manager" dialog,

#### **g. Apply Boundary Conditions (Model >> Boundaries >> Boundary Conditions)**

Apply the climate to the ground surface.

1. Select **R1** geometry by moving mouse to and clicking the region, as shown in Figure 2,
2. Click the menu of **Model >> Boundaries >> Boundary Conditions**,
3. Select the (first ) row with Y coordinate of 0 from the data grid in "**Boundary Condition**" dialog,
4. Select "**Climate**" from the "**Boundary Condition:**" drop-down list,
5. Select "DemoClimate" from the "**Climate Name:**" drop-down list,
6. Click "**OK**" button on the info message about default plot creation,
7. Click "**OK**" button to close the dialog.

**NOT  
E :**

Step 4 and 5 for Standard License of SVOOffice:  
 4. Select "Temperature Expression", or "Temperature Data" from the drop-down list,  
 5. Specify the temperature value or expression, or daily air temperature data.

Apply thermal flux at the bottom of model geometry

8. Select **R3** by moving mouse to the region 3 and clicking the region,
9. Click the menu of **Model >> Boundaries >> Boundary Conditions**,
10. Select the (2nd) row with Y coordinate of -5 from the data grid in "Boundary Conditions" dialog,
11. Select "**Flux Constant**" from the "**Boundary Conditions:**" drop-down list,
12. Enter a value in "**Constant:** " text box,  
**Constant:**  $5184 \text{ J/day-m}^2$

The estimated thermal flux on the bottom is  $0.06 \text{ w/m}^2 = 5184 \text{ J/day-m}^2$ .

13. Click "**OK**" button.

## h. Specify Initial Conditions (Model >> Initial Conditions >> Settings)

A transient model requires initial conditions.

1. Click the menu of **Model >> Initial Conditions >> Settings...**
2. Select "**Constant**" radio button,
3. Enter the initial temperature:  $13 \text{ }^{\circ}\text{C}$
4. Click "**OK**" button.

## i. Specify Output Settings (Model >> Reporting >> Plot Manager )

Specify output settings for analysis of model running results.

1. Click the menu of **Model >> Reporting >> Plot Manager**,
2. Click "**Plots**" Tab  
 In Plots tab, the default outputs are present. These can be modified if desired.
3. Select the row with the title of "**Temp**",
4. Click "**Properties...**" button, or double clicking the selected row.
5. Click "**Description**" tab,
6. Change "**Temp**" into "MyTemp" in the "Title" text box,
7. Click "**Update Method**" tab,
8. Change increment as

Increment: 1

9. Click "Output Options" Tab,
10. Check "**Write.txt File**" check box  
A text file with file name of MyTemp.txt will be created under the directory of model file after running the model.
11. Click "**OK**" button.

Specify temperature output at a particular location

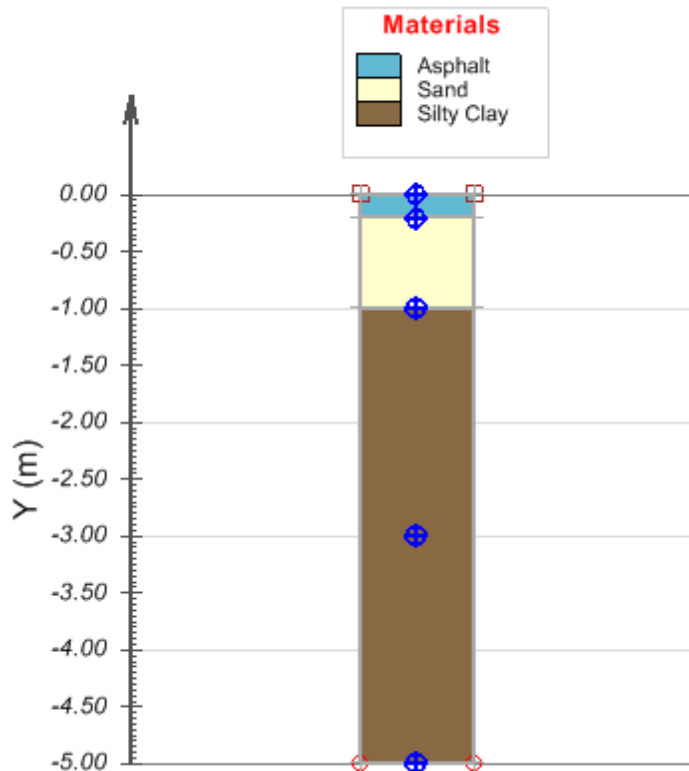
12. Click "Point" Tab,
13. Click "**Multiple Entry**" button,
14. Select the "Te" variable from the drop-down list,
15. Enter a group name in the text box,  
"**Group:**" TempAtEachLayer
16. In the data grid table, enter Y coordinate and title as following,

0	temp0
-0.2	temp02
-0.5	temp05
-1.0	temp1
-2.0	temp2
-3	temp3
-5	temp5

17. Click "**OK**" button,
18. Select all rows in "**Point**" tab,
19. Click "**Multiple Update...**" button,
20. Set increment step to 1  
Increment: 1
21. Click "**Output Options**" tab,
22. The check box Write to .txt File will be checked,  
A file TempAtEachLayer.txt will be created in the model directory after running the model.
23. Click "**OK**" button
24. Click "**OK**" button in "**Plot Manager**" dialog

Now the model settings are completed (Figure 4).





**Figure 4 Model geometry and settings.**  
The points are the location to display temperature changing

#### j. Run Model (Solve >> Analyze)

To run the model, click menu of **Solve >> Analyze**. Note that this model will take several minutes to run.

#### k. Analysis of Result

To analyze the running results, one approach is to use AcuMesh tools of SVOoffice, Another is to use the output data file that was created under the directory of the model file.

#### l. Tuning model settings

If the result is not expected, try to tune the material properties. Set a reasonable value for thermal conductivity, heat capacity, and unfrozen water content as the function of temperature, etc. Also it is very important to specify correct boundary conditions to obtain a proper solution. After a model setting is changed, it is recommended to save the model as a separate file name.

### Change climate temperature based on daily minimum and maximum temperature

**NOT**

**E :**

This feature requires the Professional License of SVOOffice2009.

In Figure 3, the yearly climate temperature change is approximated with an empirical formula. However, daily climate data may be available from a weather station. SVHeat allows user to specify the temperature based on daily average temperature or daily minimum and maximum temperature. The following instructions are to modify the model settings to use the daily min/max temperature changing pattern:

1. Save model as a separate model name,  
Click the menu of **File >> Save As**. A dialog will be opened. In this dialog, do the following:

In General Tab,

Enter the "**New File Name**": RoadPavement\_TUT1D\_MinMaxTemp,

Click **OK** button,

2. Modified climate temperature settings

Click the menu of **Model >> Boundaries >> Climate Manager ...**,

Select the row of "**DemoClimate**" from the data grid,

Double click the (2nd) column "Temperature" in the selected row to open the "Climate Properties" dialog.

Click "**Air Temperature**" Tab

Select "**Data – Smooth Cos Function**" from the "**Air Temperature Option:**" drop-down list.

Enter the following time and min/max temperature into the data grid. Here only 5 daily dataset is used for example.

Time (day)	Min Temp (C )	Max Temp (C)
0	15.6	16.2
1	14.3	27.6
2	16.2	18.7
3	14.5	16.5
4	14.8	17.0

**NOTE:**

For a large dataset, you can copy a dataset from Microsoft Excel and then paste into the data grid in the dialog.

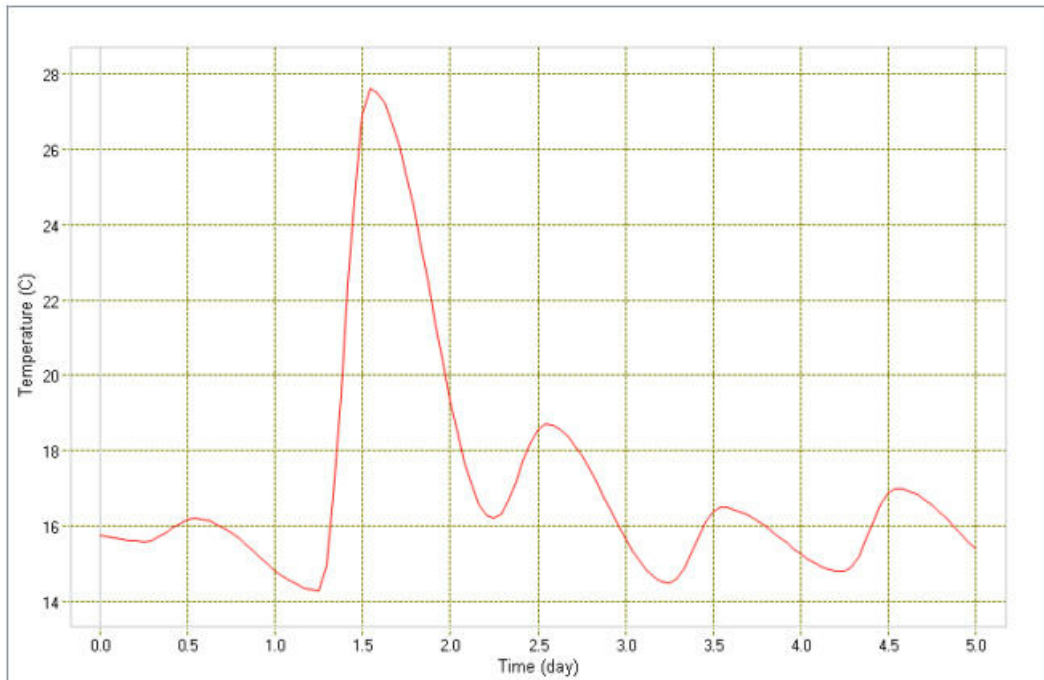
Enter the time that the daily Min/Max temperature occurs. By default, it is assumed that the minimum temperature is recorded at 6:00 am, and maximum is at 13:00 pm.

**Hour of Day at Daily Min/Max Temp**

Hours at min temp: 6

Hours at max temp : 13

Click **"Graph"** button to preview the daily temperature changing with time, as shown in Figure 8.



**Figure 8 Daily temperature changing time**

Click **"OK"** button in the graph,  
Click **"OK"** button in the **"Climate Properties"** dialog,  
Click **"OK"** button in the **"Climate Manager"** dialog.

### 3. Modify the end time of model simulation

Considering only 5 daily temperatures are input in the climate temperature, the end time of model simulation is need to change.

Click the menu **Model >> Settings**

Click **"Time"** Tab in the **"Model Settings"** dialog,

Enter the **"End Time"**: 5

Click the **"Yes"** button in the dialog suggesting to update plot specification end time automatically.

Click **"OK"** button.

### 4. Modified Output settings due to time change

Click the menu **Model >> Reporting >> Plot Manager**,

In the **Plots** Tab,

Select all rows in the data grid,

Click **"Multiple Update..."** button,

Click the **"<-"** button to set time steps for temperature output.

Click **OK** button.

In the **Point** Tab,  
Select all rows in the data grid,  
Click "**Multiple Update...**" button,  
Click the "<" button to set time steps for temperature output.  
Click **OK** button,  
Click **OK** button to close the Plot Manager.

Click the menu **Model >> Reporting >> Output Manager**,  
Click the Properties button for the AcuMeshInput.dat entry,  
On the Update Method tab set the Increment to 1,  
Click **OK** button,  
Click **OK** button to close the Output Manager.

5. Run the model  
Click the menu "**Solve >> Analyze**"

6. Analysis of results  
See above

## Change climate temperature based on hourly temperature

### **NOT E:**

This feature requires the Professional License of SVOOffice2009.

The following steps are to modify model settings so that user can specify the hourly based temperature each day.

1. Save model as a separate model name,  
Click the menu of **File >> Save As**. A dialog will be opened. In this dialog, do the following:

In General Tab,  
Enter the "**New File Name**": RoadPavement\_TUT1D\_HourlyTemp

In Time Tab.  
Select "**hr**" from the "**Time Units**" drop-down list. Click **OK** for the dialog suggesting to convert time units automatically.

Change increment and end time to simulate 1 day

Initial Increment:	0.1
Maximum Increment:	0.5
End Time:	24

Click "**OK**" button.

2. Modified climate temperature settings  
Click the menu of **Model >> Boundaries >> Climate Manager ...** ,  
Select the row of "**DemoClimate**" from the data grid,  
Double click the (2nd) column of "Temperature" in the selected row to open the "Climate Properties" dialog.  
Click "**Air Temperature**" Tab  
Select "**Data – Spline-Function**" from the "Air Temperature Options: " drop-down list,  
Enter the following time and temperature in the data grid.

**NOTE:**

You can copy dataset from Microsoft Excel and then paste to the data grid in the dialog.

Click the "**Graph...**" button to preview the temperature changing with time.  
Click "**OK**" button in the graph dialog.

Time(hr)	Temperature (C )
1	15.85
2	15.8
3	15.75
4	15.7
5	15.65
6	15.6
7	15.65
8	15.7
9	15.75
10	15.8
11	15.85
12	15.9
13	15.95
14	16
15	16.05
16	16.1
17	16.15
18	16.2
19	16.15
20	16.1
21	16.05
22	16
23	15.95
24	15.9

Click **N-Factor** Tab

Change N-Factor to 1, because the temperature at the ground surface is supposed to be equal to the air temperature.

"N-Factor Constant: " 1

Click **"OK"** button in the **"Climate Properties"** dialog.

Click **"OK"** button in the **"Climate Manager"** dialog.

3. Modified Output settings for the time unit change (**Model >> Reporting >> Plot Manager**)

Click the menu of **Model >> Reporting >> Plot Manager**,

In the **Plots** Tab,

Select all rows in the data grid,

Click **"Multiple Update..."** button,

Click the **"<-"** button to set time steps for temperature output.

Click **OK** button.

In the **Point** Tab,

Select all rows in the data grid,

Click **"Multiple Update..."** button,

Click the **"<-"** button to set time steps for temperature output.

Click **OK** button.

Click **OK** button to close the **"Plot Manager"** dialog.

Click the menu **Model >> Reporting >> Output Manager**,

Click the Properties button for the AcuMeshInput.dat entry,

On the Update Method tab set the Increment to 2,

Set the End to 24

Click **OK** button,

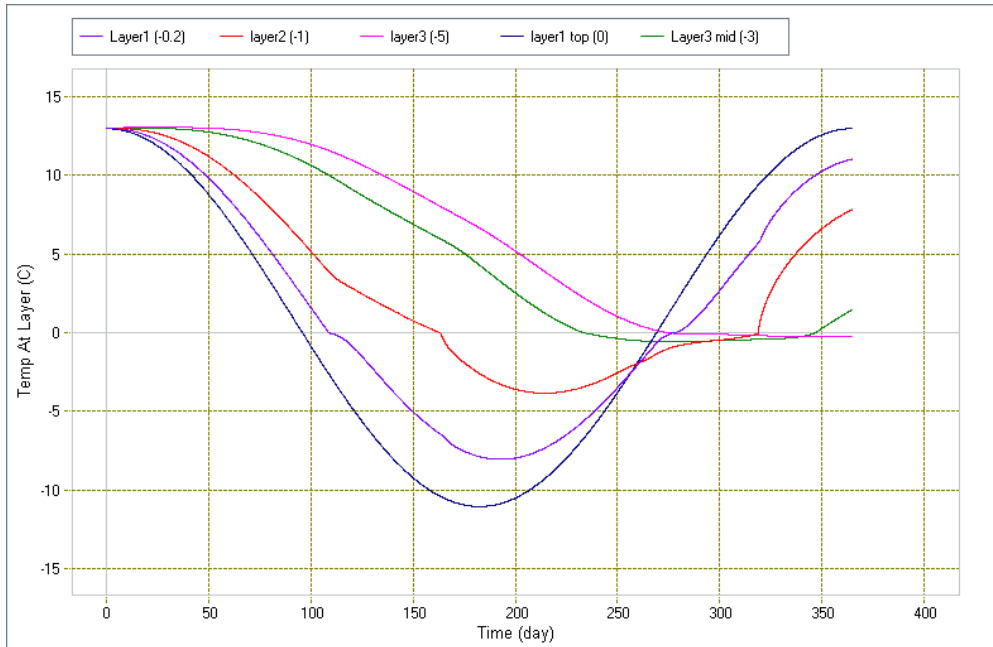
Click **OK** button to close the Output Manager.

4. Run Model  
Click the menu of **"Solve >> Analyze"**

## 3.2 Results and Discussion

### a. Display Temperature at Different Layers

1. Click menu of **Window >> AcuMesh**,
2. Click menu of **Graphs >> Plot Manager...**
3. Click **Point** tab,
4. Select the row with title of "TempAtEachLayer" from the data grid table, double click the selected row. The temperature changing at different location is illustrated in Figure 6.

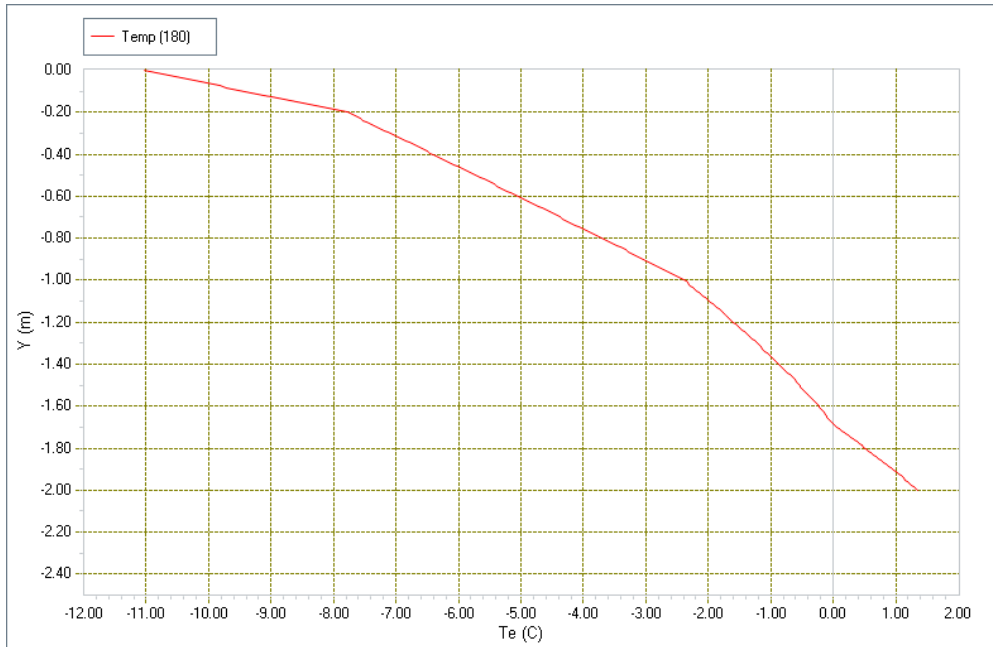


**Figure 6 Temperature distribution at different locations.**

5. Click **OK** button.

## Soil Temperature Distribution along the elevation

1. Click Plots tab,
2. Double click the row with Title of "MyTemp",
3. The temperature changing along elevation from 0 to -5 m at the time of 180 days is illustrated in Figure 7.



**Figure 7 Temperature changing along elevation at the time of day 180**

**NOTE:**

You can use the data file of TempAtEachLayer.txt that is located under the model directory to compare the simulation result with the measured temperature.

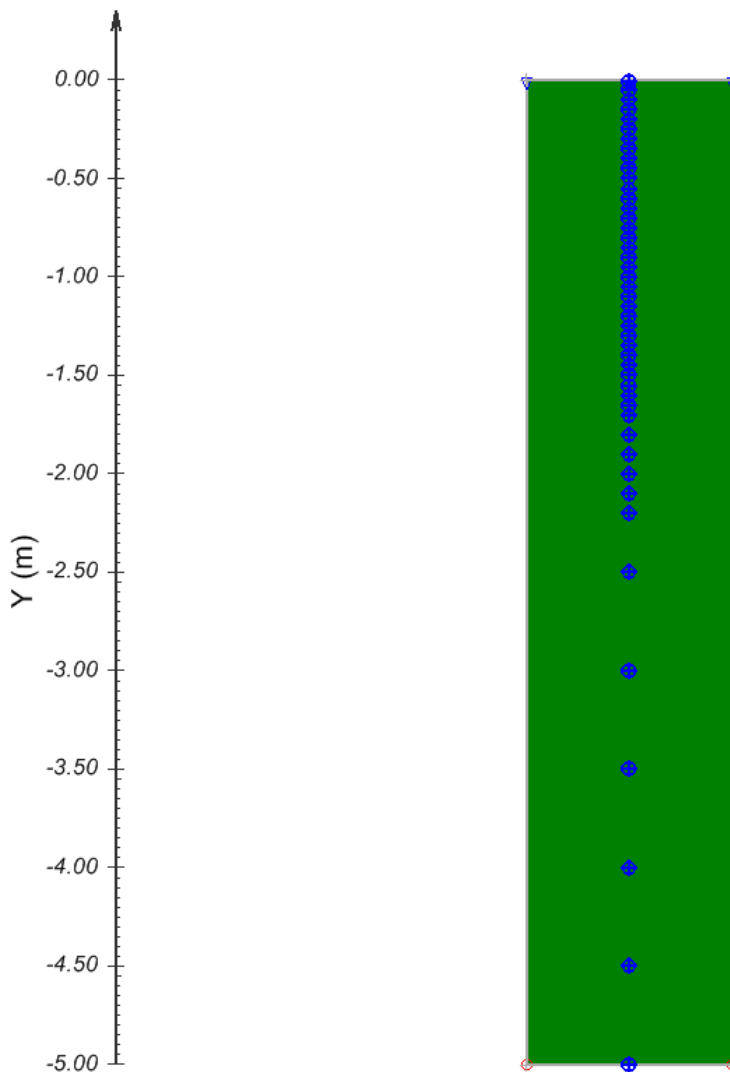


## 4 1D Freezing Front Analysis

The following example demonstrates how to setup and analyze a one dimensional column that models the freezing front as the soil undergoes a freezing and thawing cycle. The model also illustrates the distribution of soil temperature, unfrozen water content, and ice content during soil freezing and thawing. Only conductive heat flows are modeled in this example.

Project: USMEP\_Textbook  
Model: Soil\_Column\_Aldrich  
System: 1D  
Type: Transient  
Minimum authorization required: FULL

### Model Geometry



## 4.1 Model Setup

The following steps will be required to set up this model:

- a. Create model
- b. Enter geometry
- c. Specify boundary conditions
- d. Specify initial conditions
- e. Apply material properties
- f. Specify model output
- g. Run model
- h. Visualize results

### NOTE:

Any values on the dialogs there are not specifically mentioned in the steps below are assumed to be the default values currently present.

### a. Create Model

This tutorial assumes that you are already familiar with creating Projects and Models in SVOFFICE. If the "UserTutorial" project does not exist, click the *Create New Project* button located above the list of projects and then enter "UserTutorial" as a new project name.

Since FULL authorization is required for this tutorial, follow these steps to ensure full authorization is activated.

To begin creating the model in this tutorial create a new model in SVHEAT through the following steps:

1. Select a Project under which to organize the tutorial,
2. Press the *Create New SVHeat Model* button located above the list of models,
3. Enter **Freezing\_Front\_Analysis** in the Model Name text box,
4. Select the following entries:

Module:	SVHEAT
System:	1D Vertical
Type:	Transient
Units:	Metric
Time Units:	Days (day)
5. Click on the *World Coordinate System* tab,

y min = -5	y max = 0
------------	-----------
6. Move to the *Time* tab and enter the following values for time:

Start Time:	0
Initial Increment:	0.02
Maximum Increment:	0.2
End Time:	365

7. Click the *OK* button to save the model and close the *New Model* dialog,
8. The new model will be automatically added to the models list and the new model will be opened.
9. On the *Display Options* dialog, select a *Vertical Spacing* of **0.1 m** and then click *OK* to accept the settings.

### b. Enter Geometry (Model > Geometry)

This 1D column is comprised of a single material layer. The *1D Thicknesses* dialog can be used to quickly create the layer thickness:

1. Select *Model > Geometry > 1D Thicknesses...*,
2. Set the *Ground Surface (datum elevation)* to **0 m**,
3. Enter a value of **5 m** in the thicknesses in the list box and click *OK* to close the dialog.

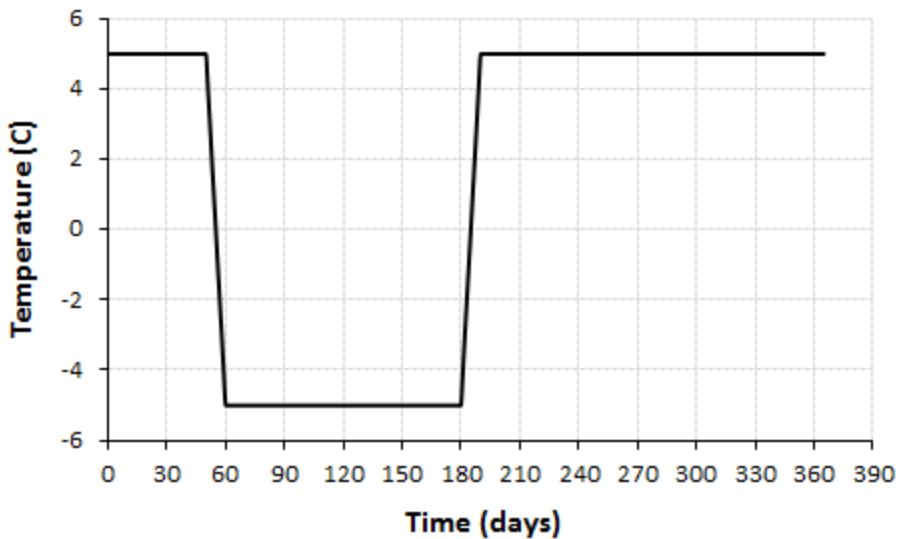
### c. Specify Boundary Conditions (Model > Boundaries)

Now that the model geometry has been defined the next step is to specify the boundary conditions. A Temperature Expression will be applied at the ground surface and a Unit Gradient boundary will be applied at the base of the column.

The steps for specifying the boundary conditions are as follows:

1. Open the *Boundary Conditions* dialog by selecting *Model > Boundaries > Boundary Conditions...* from the menu,
2. Select the *Y=0* row in the list by clicking on the row and set the *Boundary Condition* type to **Temperature Expression** using the drop down list,
3. Copy the following text and paste it in the *Expression* text box:  
$$\text{if } t < 50 \text{ then } 5 \text{ else if } t < 60 \text{ then } 5 - 1*(t-50) \text{ else if } t < 180 \text{ then } -5 \text{ else if } t < 190 \text{ then } -5 + 1*(t - 180) \text{ else } 5$$

The shape of the Temperature Expression function is shown in the figure below.
4. Select the *Y=-5* row in the list by clicking on the row and set the *Boundary Condition* type to **Flux Expression** using the drop down list,
5. Type the Y-conductivity variable name into the *Expression* text box:  
$$kty$$
6. Click *OK* to close the *Boundary Conditions* dialog.



#### d. Specify Initial Conditions (Model > Initial Conditions)

Initial conditions must be specified prior to solving a transient model. In this case we will specify an initial temperature.

1. Select *Model > Initial Conditions > Settings...* from the menu,
2. Select the *Expression* option,
3. Enter a temperature of **5 °C**,
4. Click *OK* to close the dialog.

#### e. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties. Define a material called *Jame\_1977* as follows:

1. Open the *Materials* dialog by selecting *Model > Materials > Manager...* from the menu,
2. Click the *New...* button to create a material,
3. Enter **Jame\_1977** for the material name,
4. Click *OK* and the Material Properties dialog will appear.

#### NOTE:

When a new material is created, the display color of the material can be specified using the Fill Color box in the Material Properties dialog. Any region that has a material assigned to it will display the corresponding material fill color.

5. On the *Conductivity* tab, select **Johansen** from the *Thermal Conductivity Option* drop-down,

6. Select the following settings:

Material State:	Crushed
Material Type:	Fine
Dry/Sat Conductivity:	Calculate
Solid Conductivity:	Solid Conductivity
Solid Component:	712800 J/day-m-°C
7. Click the *Volumetric Heat Capacity* tab,
8. Select the **Jame-Newman** option and enter the following values:

Soil Dry Density:	1330 kg/m <sup>3</sup>
Specific Heat Capacity of Solid Component:	837 J/kg-°C
9. Click the SFCC tab,
10. Enter the following *Phase Change Temperatures*:

From (Tef):	-0.1 °C
To (tep):	-0.607 °C
11. Set the *SFCC Method* to **Estimated by SWCC**,
12. Set the *SWCC Method* to **Fredlund and Xing** and click the *SWCC Properties...* button,
13. Enter the following predetermined *Fredlund and Xing Fit* values:

af:	127.8
nf:	1.3
mf:	2.0
hr:	500
Saturation Suction:	0.1
14. Click *OK* to close the *Fredlund and Xing Fit* dialog,
15. Move to the *VWC* tab and enter the following *Volumetric Water Content* values:

SatVWC:	0.47
VWC:	0.49
16. Click *OK* to close the *Material Properties* and *Materials Manager* dialogs.

The material will need to be applied to the model region by following these steps:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions...* from the menu,
2. For the *R1* region, select **Jame\_1977** from the Material drop down list,
3. Click *OK* to close the regions dialog.

#### f. Specify Model Output (Model > Reporting)

In this model the plots of interest are the default plots for variables such as temperature and volumetric water content. A group of *History* plots will be created separately in order to view the Temperature profiles at various locations in the column over time. The history plot

locations may be seen in the Model Geometry section as the blue circles extended through the middle of the column.

To set up these plots follow the steps below.

1. Open the *Plot Manager* dialog by selecting *Model > Reporting > Plot Manager...* from the menu,
2. Move to the *Point* tab,
3. Click the *Multiple Entry...* button,
4. Set the *Variable* to **Temperature** by selecting the value from the drop down list,
5. Set the *Plot Type* to **History** by selecting the value from the drop down list,
6. In the *Group* text box type **Soil Temperature**,
7. Copy the data from the data table at the end of this tutorial (do not include the header row) and click the *Paste* button to paste the data into the dialog,
8. Click *OK* to close the *Plot Manager* dialog.

#### g. Run Model (Solve > Analyze)

The next step is to analyze the model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model. After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the FlexPDE solver. Right-click the mouse and select "Maximize", or double-click, to enlarge any of the thumbnail plots.

These reports are intended to provide the user with low-quality graphs which give a rough indication of the results. Creating professional-quality visualizations of the results can be accomplished with ACUMESH software.

#### h. Visualize Results (Window > AcuMesh)

The visual results for the current model may be examined by selecting the *Window > ACUMESH* menu option.

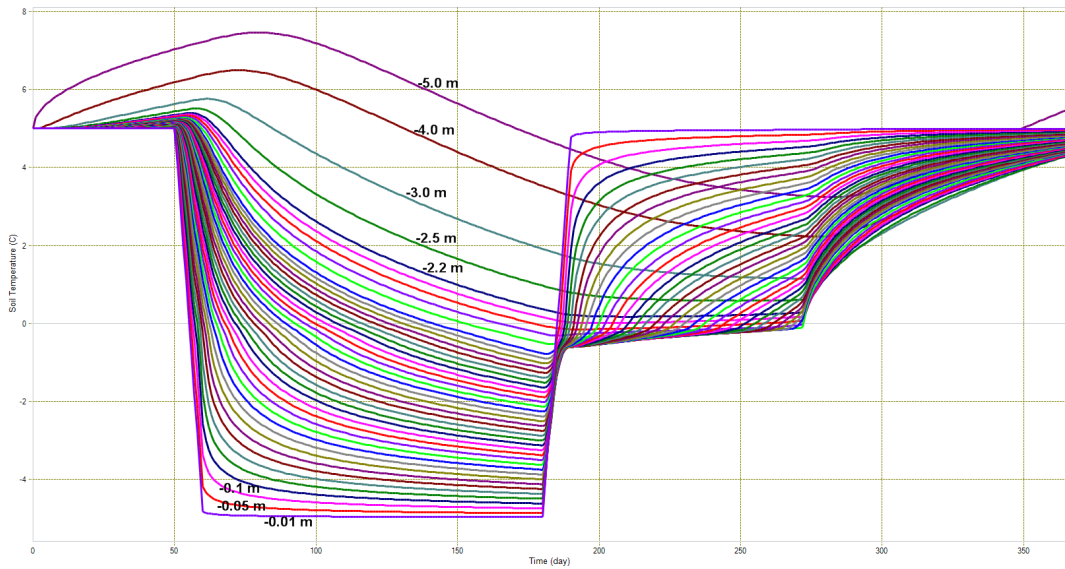
## 4.2 Results and Discussion

The history plots defined in the Model Output section provide an excellent summary of the soil temperature at various depths over time. This plot may be viewed in Acumesh by following these steps:

1. Select *Graphs > Plot Manager...* menu item,
2. Move to the *Point* tab,
3. Select the *Soil Temperature* plot from the list of plots and click the *Graph* button.

The results shown in the plot illustrate the soil temperature changes during soil freezing and thawing. In the simulation, the temperature at depth 0 m is maintained at 5 °C in the first 50 days and then it drops from 5 °C to -5 °C from day 50 to 60. After that time, the temperature holds at -5 °C. From day 180 to 190, the temperature increases from -5 °C to 5 °C, and the soil column experiences a thawing period. The history plots at positions Y=-5 m, -4 m, -3 m, -2.5 m, -2.2 m, -0.1 m, -0.05 m and -0.01 m are labeled in the screenshot

below.



## 4.3 Model Data

### History Plot Data

Y (m)	Title
-0.01	TE1
-0.05	TE2
-0.1	TE3
-0.15	TE4
-0.2	TE5
-0.25	TE6
-0.3	TE7
-0.35	TE8
-0.4	TE9
-0.45	TE10
-0.5	TE11
-0.55	TE12
-0.6	TE13
-0.65	TE14
-0.7	TE15
-0.75	TE16
-0.8	TE17
-0.85	TE18
-0.9	TE19
-0.95	TE20
-1.0	TE21

---

-1.05	TE22
-1.1	TE23
-1.15	TE24
-1.2	TE25
-1.25	TE26
-1.3	TE27
-1.35	TE28
-1.4	TE29
-1.45	TE30
-1.5	TE31
-1.55	TE32
-1.6	TE33
-1.65	TE34
-1.7	TE35
-1.8	TE36
-1.9	TE37
-2	TE38
-2.1	TE39
-2.2	TE40
-2.5	TE41
-3	TE42
-4	TE43
-5	TE44

[Return to Enter Geometry Section](#)



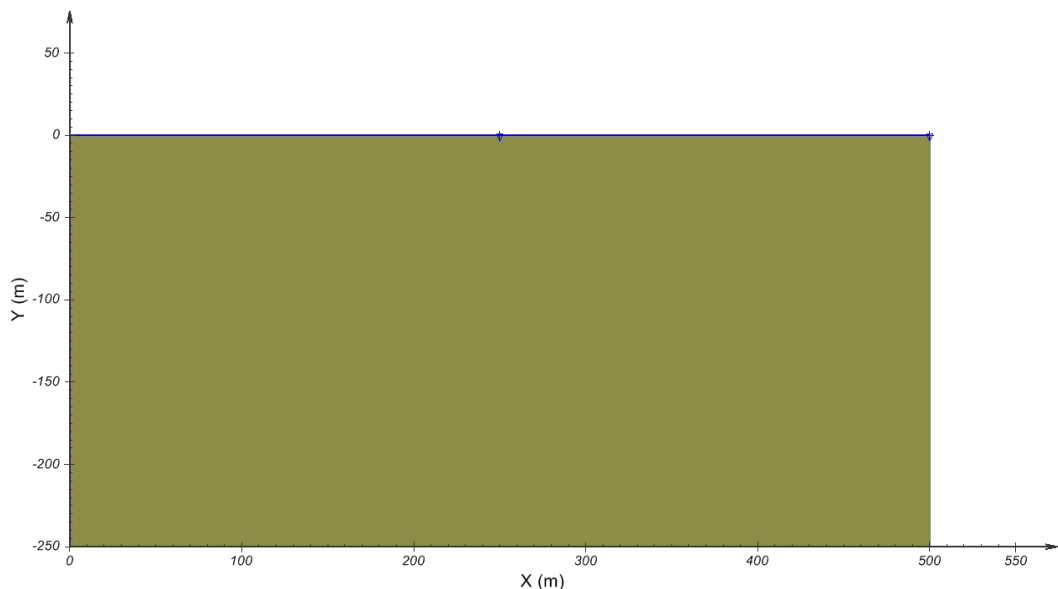
## 5 2D Partition Model

The following example demonstrates how to setup and analyze a two dimensional area that models the steady-state conditions between two adjacent areas with surface temperatures of 4 °C and -5 °C. The results from SVHEAT will be compared to the results of an analytical solution published by Harlan and Nixon (1978).

Project: USMEP\_Textbook  
Model: HarlanNixon1978  
System: 2D  
Type: Steady-State  
Minimum authorization required: Student

### Model Geometry and Description

The model geometry is composed of a single rectangular area with width 500 m and depth 250 m. The model is setup to represent two adjacent semi-infinite areas. The division between the left and right sides is the midpoint of the geometry width (250 m). The left side surface temperature is set to 4 °C and the right side surface temperature is set to -5 °C. The left side represents the inner portion of a simulated heated building and the right side represents the outdoors.



### 5.1 Model Setup

The following steps will be required to set up this model:

- Create model
- Enter geometry
- Specify boundary conditions
- Apply material properties
- Specify model output

- f. Run model
- g. Visualize results

**NOTE:**

Any values on the dialogs there are not specifically mentioned in the steps below are assumed to be the default values currently present.

### a. Create Model

This tutorial assumes that you are already familiar with creating Projects and Models in SVOFFICE. If the "UserTutorial" project does not exist, click the *Create New Project* button located above the list of projects and then enter "UserTutorial" as a new project name.

To begin creating the model in this tutorial create a new model in SVHEAT through the following steps:

1. Select a Project under which to organize the tutorial,
2. Press the *Create New SVHeat Model* button located above the list of models,
3. Enter **2D\_Partition\_Model** in the Model Name text box,
4. Select the following entries:

Module:	SVHEAT
System:	2D
Type:	Steady-State
Units:	Metric
Time Units:	s
5. Click on the *World Coordinate System* tab and enter the values shown below:

x min = 0	x max = 500
y min = -250	y max = 50
6. Click the *OK* button to save the model and close the *New Model* dialog,
7. The new model will be automatically added to the models list and the new model will be opened.
8. On the *Display Options* dialog, click *OK* to accept the default settings.

### b. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions and can be either drawn by the user or defined as a set of coordinates. Model geometry can also be imported from either .DXF files or from existing models.

This model consists of a single region. To add the region follow these steps:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions...* from the menu,
2. Change the region name from *R1* to *Ground*. To do this, highlight the name and type the text,

The shapes that define each material region will now be created. The steps to create the *Ground* region are as follows:

1. Click on the *Ground* region item in the region list box and press the *Properties...* button,
2. Click on the *New Polygon...* button to open the *New Polygon Shape* dialog,
3. Copy and paste the region coordinates from the table below into the dialog using the *Paste Points* button (do not copy the header row),
4. Press *OK* to close the dialog.

**Region: Ground**

X (m)	Y (m)
0	0
0	-250
500	-250
500	0
250	0

**c. Specify Boundary Conditions (Model > Boundaries)**

Now that the model geometry has been defined, the next step is to specify the boundary conditions. A temperature of 4 °C will be applied to the ground surface on the left side of the model and a temperature of -5 °C will be applied to the ground surface on the right side of the model. By default, a No BC boundary condition is applied to the remainder of the model. The steps for specifying the boundary conditions are as follows:

1. Open the *Boundary Conditions* dialog by selecting *Model > Boundaries > Boundary Conditions...* from the menu,
2. Select the point (500,0) from the list,
3. From the Boundary Condition drop-down select a **Temperature Constant** boundary condition. This will cause the Constant box to be enabled,
4. In the Constant box enter a temperature of **-5 °C**,
5. Select the point (250,0) from the list,
6. From the Boundary Condition drop-down select a **Temperature Constant** boundary condition.
7. In the Constant box enter a temperature of **4 °C**,
8. Click *OK* to save the Boundary Conditions and return to the workspace.

**d. Apply Material Properties (Model > Materials)**

The next step in defining the model is to enter the material properties. Define a material called *Soil* as follows:

1. Open the *Materials Manager* dialog by selecting *Model > Materials > Manager...* from the menu,

2. Click the *New...* button to create a material,
3. Enter *Soil* for the material name,
4. Click *OK* and the *Material Properties* dialog will appear.

**NOTE:**

When a new material is created, the display color of the material can be specified using the Fill Color box in the Material Properties dialog. Any region that has a material assigned to it will display the corresponding material fill color.

5. On the *Conductivity* tab, select **Data** from the *Thermal Conductivity Option* drop-down,
6. Click the *Data...* button to open the *Thermal Conductivity Data* dialog,
7. Select the **Data Changes with Temperature** option,
8. Enter the data as shown below into the data table on the dialog,

Te (°C)	k (J/s-m-°C)
-5	1
4	1

9. Click *OK* to close the *Thermal Conductivity Data* dialog,
10. Move to the *Volumetric Heat Capacity* tab,
11. Select the **Constant** option and enter the following values:
 

Unfrozen Volumetric Heat Capacity: 1,950,000 J/m<sup>3</sup>-°C  
 Frozen Volumetric Heat Capacity: 1,950,000 J/m<sup>3</sup>-°C
12. Move to the *SFCC* tab,
13. Enter the following *Phase Change Temperatures*:
 

From (Tef): -0.01 °C  
 To (Tep): -0.5 °C
14. Set the *SFCC Method* to **None** using the drop down,
15. Move to the *VWC* tab and enter the following *Volumetric Water Content* values:
 

SatVWC: 0.35  
 VWC: 0.35
16. Click *OK* to close the *Material Properties* and *Materials Manager* dialogs.

The material will need to be applied to the model region by following these steps:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions...* from the menu,
2. For the *Ground* region, select **Soil** from the Material drop down list,
3. Click *OK* to close the regions dialog.

### e. Specify Model Output (Model > Reporting)

In this model the plots of interest are the temperature throughout the model. For demonstration purposes the temperature along a cross-section of the model will also be plotted. This section covers how the user may output these plots.

1. Open the *Plot Manager* dialog by selecting *Model > Reporting > Plot Manager...* from the menu,
2. On the *Plots* tab select the **Add New Elevation Plot** button located in the *Add New Plot* group to open the *Plot Properties* dialog,
3. On the *Description* tab, Enter a Title of **Temp along Y=-125**,
4. Select the **Temperature** variable from the drop down list,
5. Move to the *Range* tab and enter the following coordinates:  
X1: 0            Y1: -125  
X2: 500        Y2: -125
6. Move to the *Output* tab and check the **Write .txt File** check box so that the plot is viewable in the Acumesh software,
7. Click *OK* to close the *Plot Properties - Plot* dialog and *Plot Manager* dialog.

The most basic plots have now been defined. As the user becomes familiar with the software additional plots may be created and customized.

### f. Run Model (Solve > Analyze)

The next step is to analyze the model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model. After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the FlexPDE solver. Right-click the mouse and select "Maximize", or double-click, to enlarge any of the thumbnail plots.

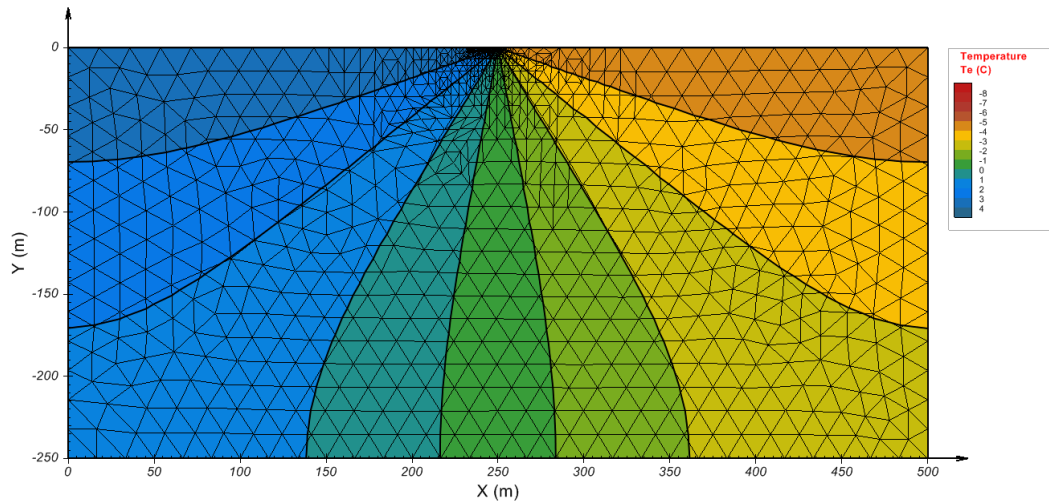
These reports are intended to provide the user with low-quality graphs which give a rough indication of the results. Creating professional-quality visualizations of the results can be accomplished with ACUMESH software.

### g. Visualize Results (Window > AcuMesh)

The visual results for the current model may be examined by selecting the *Window > ACUMESH* menu option.

## 5.2 Results and Discussion

The default plot that appears in Acumesh is a contour plot of the Temperature variable. The finite element mesh used to solve the model is also displayed by default. The effect of automatic mesh refinement can be seen at the midpoint on the ground surface where the temperature value changes from 4 °C to -5 °C.



- Analytical Solution

Figure 1 and Figure 2 below show the temperature results as produced by SVHeat and the analytical solution, respectively. The solutions are in agreement with respect to the location of the freezing front as well as the remaining temperature contours in both the frozen and thawed portions of the material.

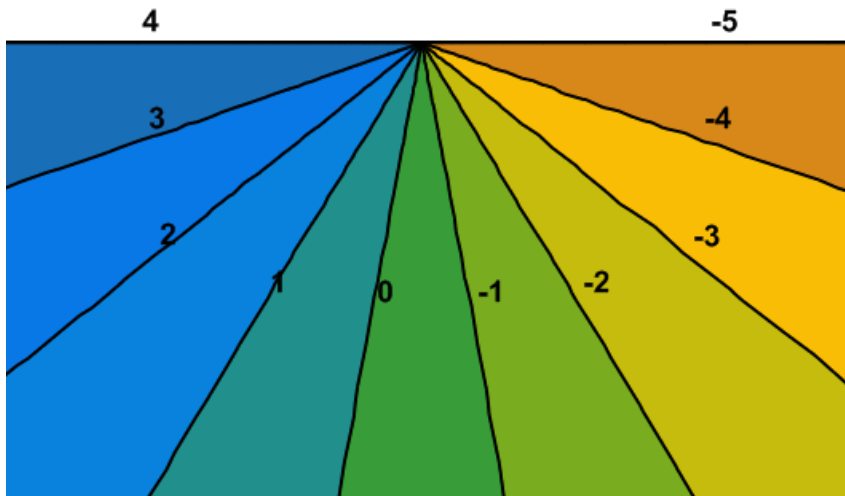
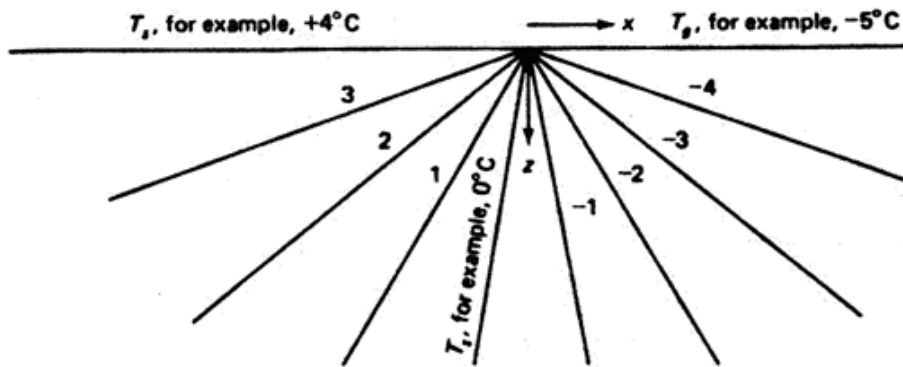


Figure 1: SVHEAT Temperature contours



**Figure 2:** Analytical temperature contours (Harlan and Nixon 1978)

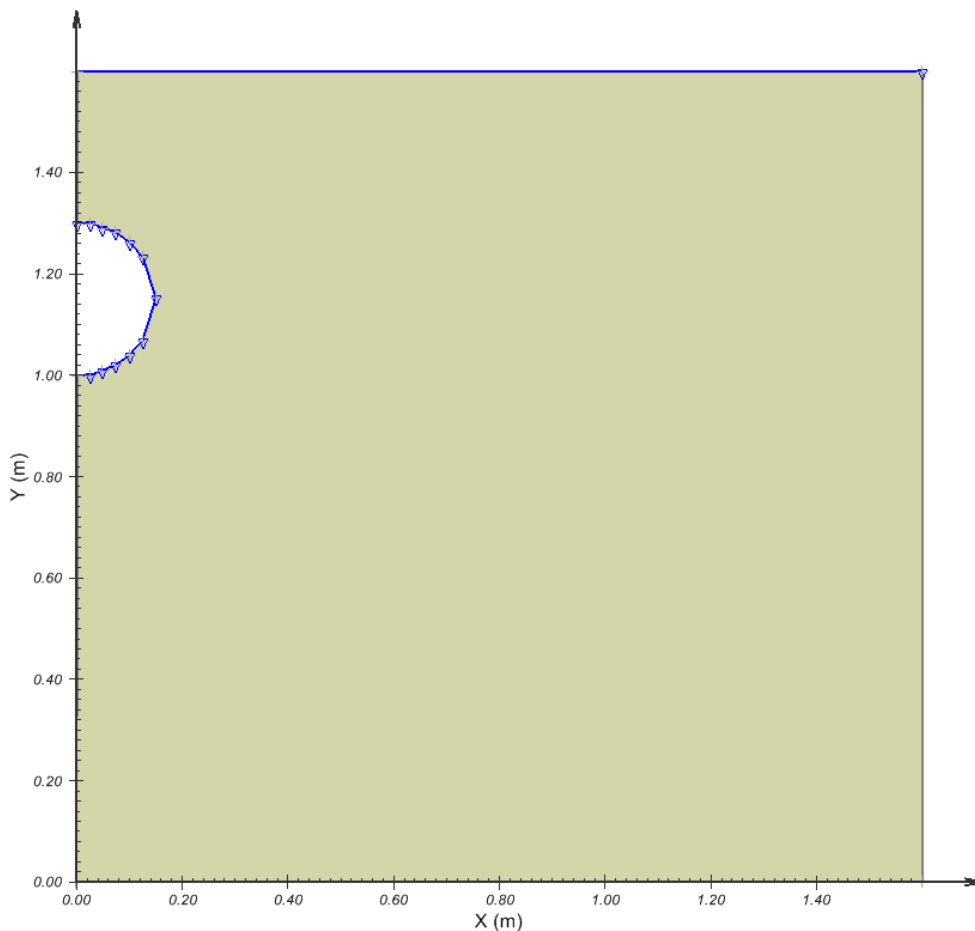
## 6 2D Chilled Pipe

The following example demonstrates how to setup and analyze a two dimensional area that models the transient conditions of a chilled pipeline buried in an area of discontinuous permafrost. This model was originally published by Coutts and Konrad (1994).

Project: USMEP\_Textbook  
Model: CouttsKonrad  
System: 2D  
Type: Transient  
Minimum authorization required: Full

### Model Geometry and Description

The model geometry is composed of a single square area of length 1.5 m with a half pipe of radius 0.15 m carved out of the left side of the square. The pipeline is chilled to a temperature of  $-2^{\circ}\text{C}$ . The soil surrounding the pipe has an initial temperature of  $3^{\circ}\text{C}$ . The soil surface temperature is kept constant at  $3^{\circ}\text{C}$ .





## 6.1 Model Setup

The following steps will be required to set up this model:

- a. Create model
- b. Enter geometry
- c. Specify boundary conditions
- d. Specify initial conditions
- e. Apply material properties
- f. Run model
- g. Visualize results

### NOTE:

Any values on the dialogs there are not specifically mentioned in the steps below are assumed to be the default values currently present.

### a. Create Model

This tutorial assumes that you are already familiar with creating Projects and Models in SVOFFICE. If the "UserTutorial" project does not exist, click the *Create New Project* button located above the list of projects and then enter "UserTutorial" as a new project name.

Since FULL authorization is required for this tutorial, follow these steps to ensure full authorization is activated.

To begin creating the model in this tutorial create a new model in SVHEAT through the following steps:

1. Select a Project under which to organize the tutorial,
2. Press the *Create New SVHeat Model* button located above the list of models,
3. Enter **2D\_Chilled\_Pipe** in the Model Name text box,
4. Select the following entries:

Module:	SVHEAT
System:	2D
Type:	Transient
Units:	Metric
Time Units:	Days (day)
5. Click on the *World Coordinate System* tab and enter the values below:

x min = 0	x max = 1.6
y min = 0	y max = 1.6
6. Move to the *Time* tab and enter the following values for time:

Start Time:	0
Initial Increment:	0.1
Maximum Increment:	1
End Time:	730

7. Click the *OK* button to save the model and close the *New Model* dialog,
8. The new model will be automatically added to the models list and the new model will be opened.
9. On the *Display Options* dialog, select a *Horizontal and Vertical Spacing* of **0.1 m** and then click *OK* to accept the settings.

## b. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions and can be either drawn by the user or defined as a set of coordinates. Model geometry can also be imported from either .DXF files or from existing models.

This model consists of a single region. To add the region follow these steps:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions...* from the menu,
2. Change the region name from *R1* to *Ground*. To do this, highlight the name and type the text,

The shapes that define each material region will now be created. The steps to create the *Ground* region are as follows:

1. Click on the *Ground* region item in the region list box and press the *Properties...* button,
2. Click on the *New Polygon...* button to open the *New Polygon Shape* dialog,
3. Copy and paste the region coordinates from the table below into the dialog using the *Paste Points* button (do not copy the header row),
4. Press *OK* to close the dialogs.

### Region: Ground

X (m)	Y (m)
0.001	1
0	0
1.6	0
1.6	1.6
0.001	1.6
0.001	1.3
0.027	1.3
0.049	1.291
0.074	1.282
0.1	1.262
0.125	1.233
0.15	1.152
0.125	1.069
0.1	1.04
0.074	1.02
0.049	1.01
0.027	1

### c. Specify Boundary Conditions (Model > Boundaries)

Now that the model geometry has been defined, the next step is to specify the boundary conditions. The pipeline boundary condition is set to a constant temperature of -2 °C. The *Ground* surface boundary temperature is set to a constant of 3 °C. By default, a No BC boundary condition is applied to the remainder of the model. The steps for specifying these boundary conditions are as follows:

1. Open the *Boundary Conditions* dialog by selecting *Model > Boundaries > Boundary Conditions...* from the menu,
2. Select the point (1.6,1.6) from the list,
3. From the Boundary Condition drop-down select a **Temperature Constant** boundary condition. This will cause the Constant box to be enabled,
4. In the Constant box enter a temperature of **3 °C**,
5. Select the point (0.001,1.3) from the list,
6. From the Boundary Condition drop-down select a **Temperature Constant** boundary condition.
7. In the Constant box enter a temperature of **-2 °C**,
8. Select the next point (0.027,1.3) and every subsequent point in the list by dragging the mouse over all of these rows,
9. Assign a boundary condition type of **Continue** to the selected rows,
10. Click *OK* to save the Boundary Conditions and return to the workspace.

### d. Specify Initial Conditions (Model > Initial Conditions)

Initial conditions must be specified prior to solving a transient model. In this case we will specify an initial temperature.

1. Select *Model > Initial Conditions > Settings...* from the menu,
2. Select the **Constant** option,
3. Enter a temperature of **3 °C**,
4. Click *OK* to close the dialog.

#### e. Apply Material Properties (*Model > Materials*)

The next step in defining the model is to enter the material properties. Define a material called *Soil* as follows:

1. Open the *Materials Manager* dialog by selecting *Model > Materials > Manager...* from the menu,
2. Click the *New...* button to create a material,
3. Enter *Soil* for the material name,
4. Click *OK* and the *Material Properties* dialog will appear.

#### NOTE:

When a new material is created, the display color of the material can be specified using the Fill Color box in the Material Properties dialog. Any region that has a material assigned to it will display the corresponding material fill color.

5. On the *Conductivity* tab, select **Constant** from the *Thermal Conductivity Option* drop-down,
6. Uncheck the *Same value for frozen or unfrozen material* check box and enter the following *Thermal Conductivity* values:
 

Unfrozen Material:	129,600 J/day-m-°C
Frozen Material:	155,520 J/day-m-°C
7. Click the *Data...* button to open the *Thermal Conductivity Data* dialog,
8. Move to the *Volumetric Heat Capacity* tab,
9. Select the **Constant** option and enter the following values:
 

Unfrozen Volumetric Heat Capacity:	2,500,000 J/m <sup>3</sup> -°C
Frozen Volumetric Heat Capacity:	2,070,000 J/m <sup>3</sup> -°C
10. Move to the the *SFCC* tab,
11. Enter the following *Phase Change Temperatures*:
 

From (Tef):	-0.01 °C
To (tep):	-0.5 °C
12. Set the *SFCC Method* to **Multi-Linear Estimation** using the drop down,
13. Set the *Residual Unfrozen Water Content* value to **0**,
14. Move to the *VWC* tab and enter the following *Volumetric Water Content* values:

SatVWC: 0.35

VWC: 0.35

15. Click *OK* to close the *Material Properties* and *Materials Manager* dialogs.

The material will need to be applied to the model region by following these steps:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions...* from the menu,
2. For the *Ground* region, select **Soil** from the Material drop down list,
3. Click *OK* to close the regions dialog.

#### f. Run Model (Solve > Analyze)

The next step is to analyze the model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model. After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the FlexPDE solver. Right-click the mouse and select "Maximize", or double-click, to enlarge any of the thumbnail plots.

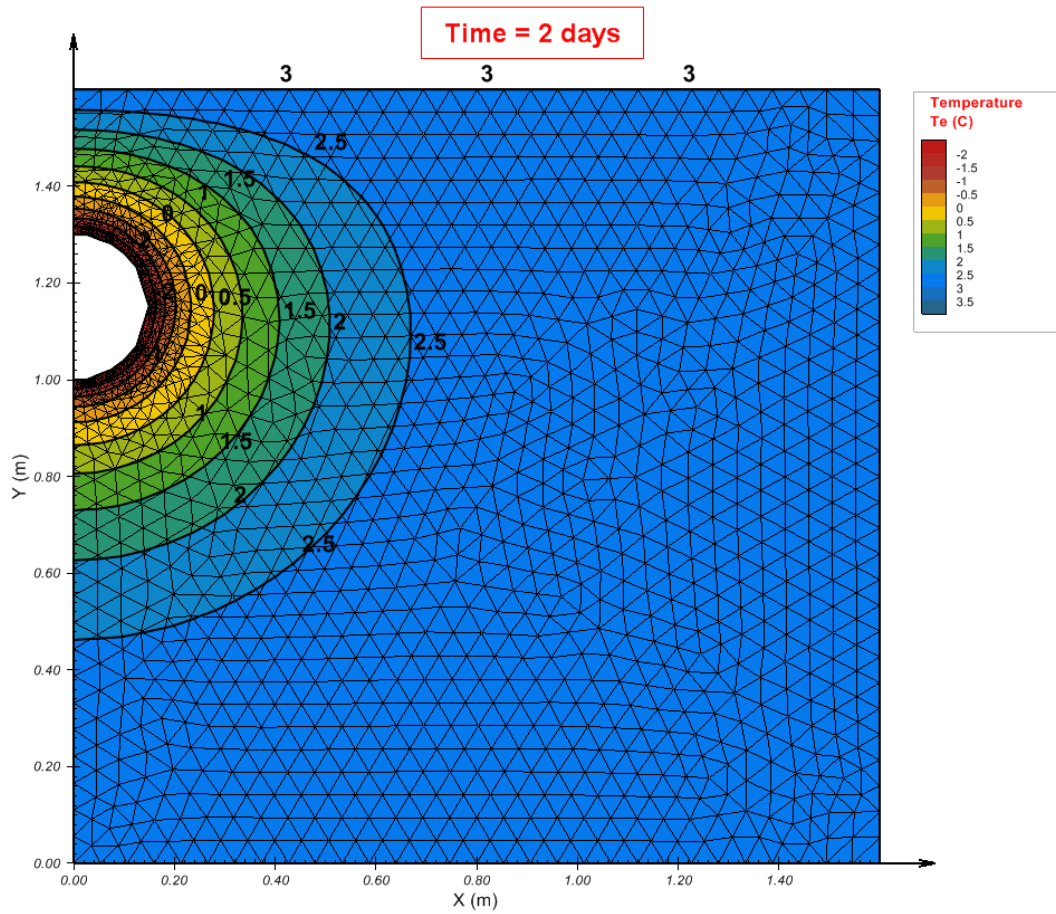
These reports are intended to provide the user with low-quality graphs which give a rough indication of the results. Creating professional-quality visualizations of the results can be accomplished with ACUMESH software.

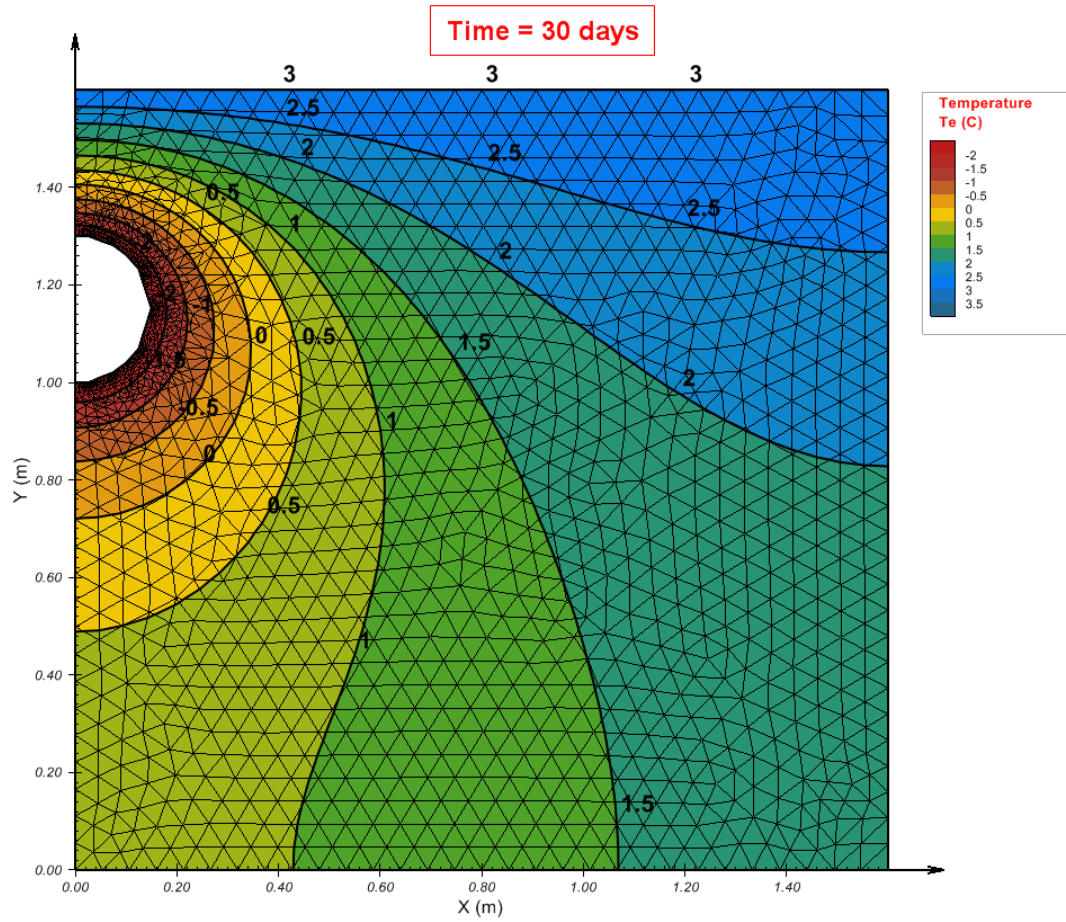
#### g. Visualize Results (Window > AcuMesh)

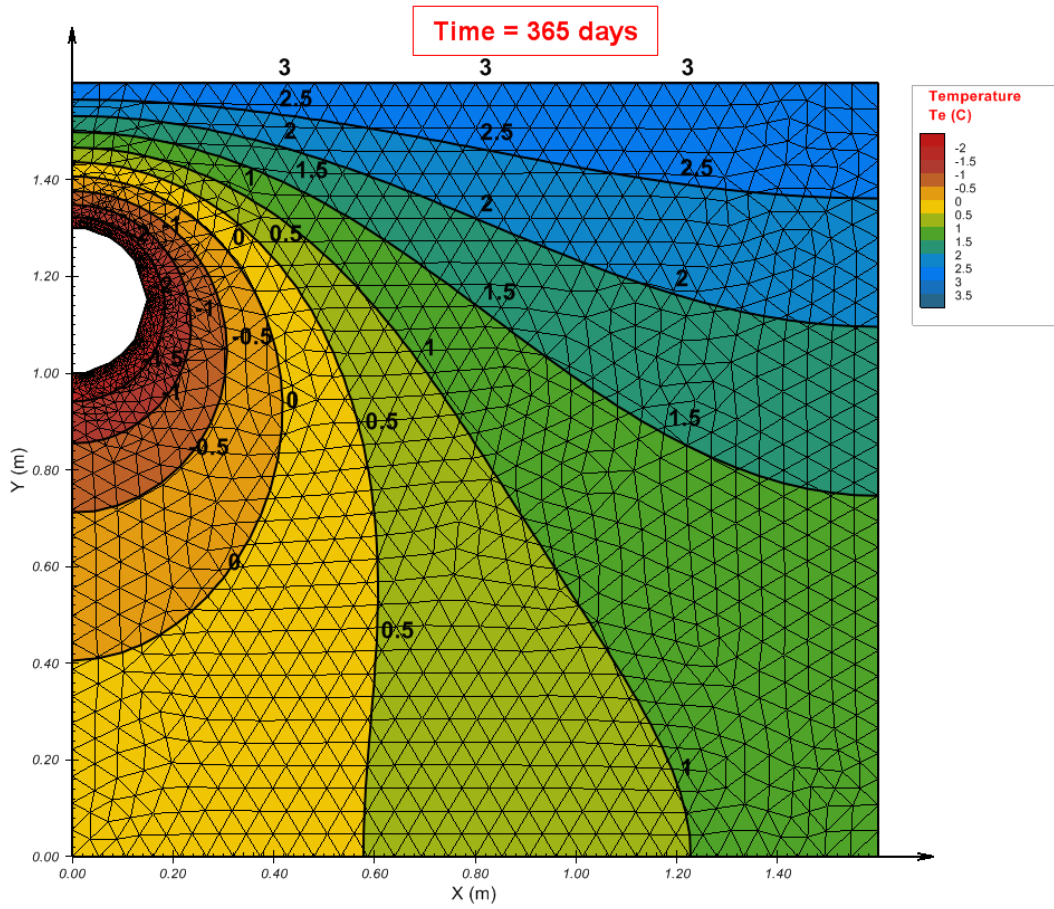
The visual results for the current model may be examined by selecting the *Window > ACUMESH* menu option.

## 6.2 Results and Discussion

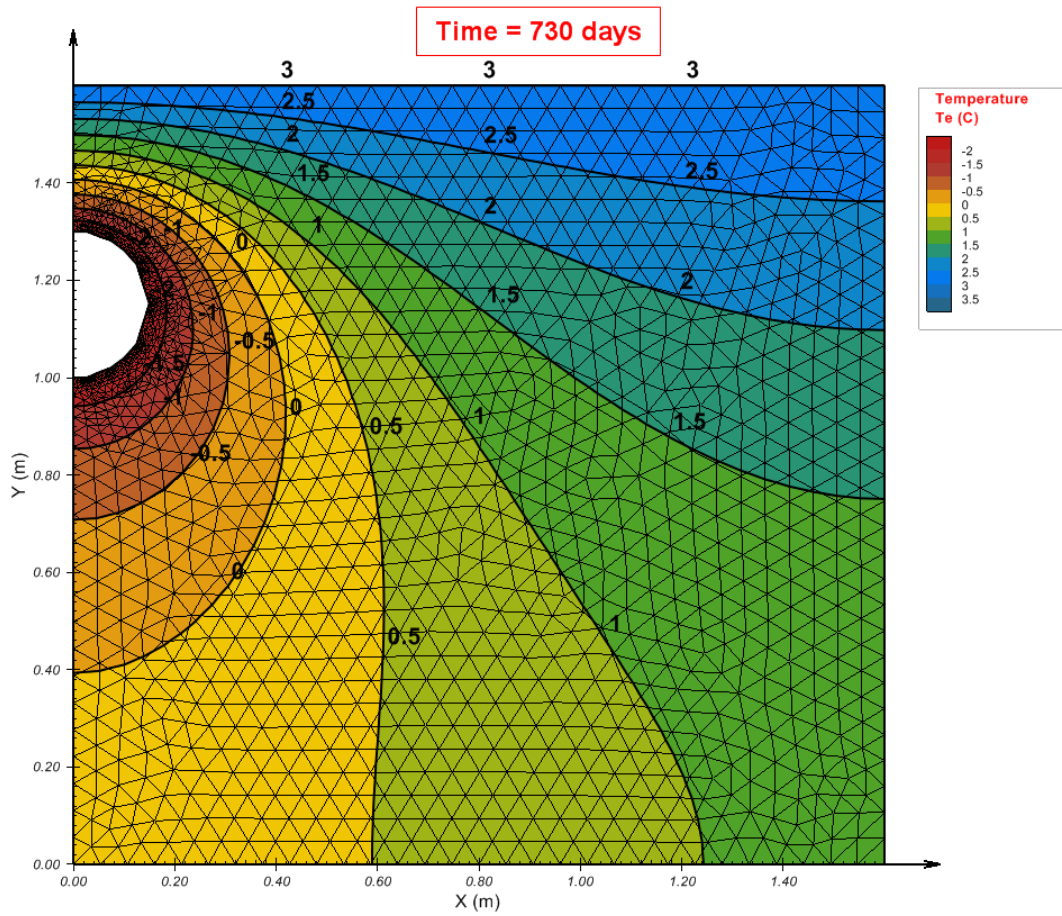
The default plot that appears in Acumesh is a contour plot of the Temperature variable. The finite element mesh used to solve the model is also displayed by default. The selected time step may be changed by using the drop down in the workspace toolbar. The screenshots below show the progression of the freezing front at the time steps 2 days, 30 days, 365 days, and 730 days.









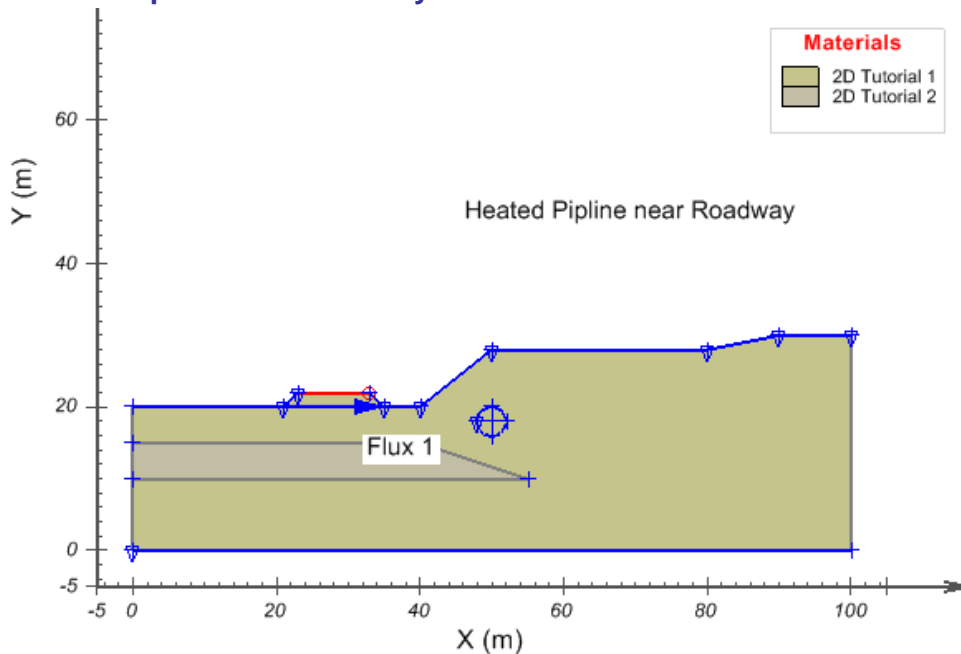


## 7 2D Heated Pipeline

The following example introduces some of the basic features of SVHEAT and sets up a model of a buried pipeline. The purpose of this model is to determine the effects of the heated pipeline on the frozen ground and the nearby roadway. This steady-state model is composed of two regions and two materials. The model data and material properties are provided below.

Project: Roadway  
Model: PipeLineTut2D  
Minimum authorization required: STUDENT

### Model Description and Geometry



Slope Region

Seam Region

Shape 1 - polygon

Shape 2 -Circle

X	Y		X	Y		X	Y
0	0	Center:	50	18		0	10
100	0					55	10
100	30	Radius:	2			40	15
90	30					0	15
80	28						
50	28						
40	20						
35	20						
33	22						
23	22						
21	20						

0	20							
0	15							
0	10							

## Material Properties

The only soil properties required to solve this problem are those related to thermal conductivity since a steady-state analysis is being performed.

Material 1 - Slope Region:

Thermal Conductivity Data:

Temperature (°C)	Conductivity (J/hr-m- °C)
-10	5688
-1	5652
-0.1	5616
0	5148
0.1	4680
1	4644
10	4608

Material 2 - Seam Region:

Thermal Conductivity Data:

Temperature (°C)	Conductivity (J/hr-m- °C)
-10	7200
-1	6840
-0.1	6480
0	5400
0.1	4680
1	4500
10	4320

## 7.1 Model Setup

The following steps are required in order to set up and solve the model described in the preceding section. The steps fall under the general categories of:

- Create model
- Enter geometry
- Specify boundary conditions
- Apply material properties
- Specify model output
- Run model
- Visualize results

### NOTE:

Any values on the dialogs there are not specifically mentioned in the steps below are

assumed to be the default values currently present.

### a. Create Model

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Create a new project called "UserTutorial" by pressing the *New* button next to the list of projects,
3. Create a new model called "Heated\_Pipeline" in the *Create New Model* dialog. Use the settings shown below when creating the new model,
4. Select the following:

Application:	SVHEAT
System:	2D
Type:	Steady-State
Units:	Metric
Time Units:	Hours (hr)
5. Click on the *World Coordinate System* tab by selecting the *World Coordinate System* tab on the *Create New Model* dialog,
6. Enter the World Coordinates System coordinates shown below into the dialog,

x min = -5	x max = 105
y min = -5	y max = 40
7. Click *OK* to close the dialog.

The workspace grid spacing can be set to aid in defining region shapes. The geometry in this model only has a precision of 1m.

1. On the Grid Spacing dialog enter 1 for both the horizontal and vertical spacing, and
2. Click *OK* to close the dialog.

### b. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

This model is divided into two regions; namely the Slope region and the Seam region. Use the following steps to add the necessary regions:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions* from the menu,
2. Change the first region name from Region 1 (i.e., R1) to Slope. To do this, highlight the name and type the new name Slope,
3. Press the *New* button to add a second region,

4. Change the name of the second region to Seam, and
5. Click *OK* to close the dialog.

Note that when drawing a geometric shape, information will be added to the region that is current in the Region Selector. The Region Selector is found at the top of the workspace and appears in the following image:



- **Define the Slope Region – Shape 1**

Shape 1 refers to the entire region consisting of material 1.

1. Select "Slope" as the region in the Region Selector,
2. Select *Draw > Geometry > Region Polygon* from the menu,
3. The cursor will now be changed to cross hairs,
4. Move the cursor near (0,0) in the drawing space. You can view the coordinates of the current position of the mouse in the status bar just below the drawing space,
5. To select the point as part of the shape, left click on the point,
6. Now move the cursor near (100,0). Left click on the point. A line is now drawn from (0,0) to (100,0),
7. Repeat this process for the remaining points as provided at the beginning of this tutorial. This will define the geometry of the Slope Region,
8. For the last point (0,10), double-click on the point to finish the shape. A line is now drawn from (0,15) to (0,10) and the shape is automatically finished. SVHEAT will draw a line from (0,10) back to the starting point, (0,0).

- **Define the Slope Region – Shape 2**

Shape 2 refers to the geometry of the heated pipeline.

9. Ensure the Slope region is current in the region selector,
10. Select *Draw > Geometry > Region Circle* from the menu,
11. The cursor will now be changed to cross hairs,
12. Move the cursor near (50,18) in the drawing space. You can view the coordinates of the current position of the mouse in the status bar just below the drawing space,
13. To select the point as the circle center left click on the desired point,
14. Drag the cursor out to a *radius* of 2,
15. Left click to finish the circle.

**NOTE:**

If a mistake was made entering the coordinate points for a shape. Select a shape with the mouse and select *Edit > Delete* from the menu. This will remove the entire shape from the region. To edit the shape use the *Region Properties* dialog.

- **Define the Seam Region**

The Seam Region corresponds to material 2 in the layout of the example,

16. Ensure that Seam is current in the region selector,
17. Select *Draw > Geometry > Region Polygon* from the menu,
18. The cursor will now be changed to cross hairs,
19. Move the cursor near (0,10) in the drawing space. You can view the coordinates of the current position of the mouse in the status bar just below the drawing space,
20. To select the point as part of the shape left click on the point,
21. Now move the cursor near (55,10). Left click on the point. A line is now drawn from (0,10) to (55,10),
22. Now move the cursor near (40,15). Left click on the point. A line is now drawn from (55,10) to (40,15), and
23. For the *last point*(0,15), left click to snap the cursor to the point. Double-click on the point to finish the shape. A line is now drawn from (40,15) to (0,15) and the shape of the seam is automatically finished. SVHEAT will draw a line from (0,15) back to the start point, (0,10).

The diagram will appear as shown at the beginning of this tutorial after all the region geometries have been entered.

### c. Specify Boundary Conditions (Model > Boundaries)

Boundary conditions must be applied to the points associated with each region. Once a boundary condition is applied to a boundary point, this defines the starting point for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until they are re-defined. The user may not define two different boundary conditions to the same line segment.

More information on boundary conditions can be found in *Menu System > Model Menu > Boundary Conditions > 2D Boundary Conditions* in your User's Manual.

Now that all of the regions of the model geometry have been successfully defined, the next step is to specify the boundary conditions. An approximate geothermal gradient of 1°C/30m will be simulated by setting the temperature at the ground surface to -6°C and the surface at the base of the model to -5°C. The temperature of the pipe is 9°C and the heat generated by the warming of the roadway is set as 200 kJ/hr (i.e., a thermal flux boundary condition). The steps associated with specifying the boundary conditions are as follows:

- **Slope Region – Shape 1**

1. Select the Slope region in the Region Selector,
2. From the menu select *Model > Boundaries > Boundary Conditions*. The

- boundary conditions dialog will open,
3. Select point (0,0) from the list,
  4. From the *Boundary Condition* drop-down menu, select a "Temperature Constant" boundary condition. This will cause the Constant box to be enabled,
  5. Enter a temperature of -5,
  6. Select the point (100,0) from the list,
  7. From the Boundary Condition drop-down select a "Zero Flux" boundary condition,
  8. Select points (100,30) through (35,20) from the list,
  9. From the *Boundary Condition* drop-down menu, select a "Temperature Constant" boundary condition. This will cause the Constant box to be enabled,
  10. Enter a temperature of -6,
  11. Repeat for the remaining points referring to this table:

X	Y	Boundary Condition	Expression
0	0	Temperature Constant	-5
100	0	Zero Flux	
100	30	Temperature Constant	-6
90	30	Zero Flux	
80	28	Continue	
50	28	Continue	
40	20	Continue	
35	20	Continue	
33	22	Flux Constant	7200 (J/hr/m <sup>2</sup> )
23	22	Temperature Constant	-6
21	20	Continue	
0	20	Zero Flux	
0	15	Continue	
0	10	Continue	

**NOTE:**

The Temperature Constant boundary condition for the point (100,30) becomes the boundary condition for the following line segments that have a Continue boundary condition until a new boundary condition is specified. By specifying a Flux Constant condition at point (33,22) the Continue boundary condition is stopped.

**NOTE:**

The difference between the No BC boundary condition type and the Zero Flux boundary condition type is one of placement. These two boundary condition types are interchangeable on exterior boundaries of a model and result in zero flow across the boundary. However, on interior boundaries of a model the No BC type is the default boundary condition and results in no effect on the flux across a boundary. The Zero Flow boundary condition type when applied to an internal boundary of a model results in zero flux across that boundary, i.e., a Zero Flux type is effectively a barrier that prevents any heat flow from crossing the boundary in the case of SVHeat.

- **Slope Region – Shape 2**

12. Select Shape 2,
13. In the Boundary Condition drop down select a Temperature Constant boundary condition,
14. Enter the value 9, and
15. Click *OK* to save the input Boundary Conditions and return to the workspace.

#### **d. Apply Material Properties (Model > Materials)**

The next step in defining the model is to enter the material properties for the two materials that are used for the model. A material called *2D Tutorial 1* is defined for the major material region (i.e., Slope Region) and the material called *2D Tutorial 2* is defined for the smaller material region (i.e., Seam Region). This section will provide instructions on creating the first material. Repeat the process to add the other material.

1. Open the *Materials* dialog by selecting *Model > Materials > Manager* from the menu,
2. Click the *New...* button to create a material,
3. Enter *2D Tutorial 1* for the material name,
4. Click *OK* and the Material Properties dialog will appear,

#### **NOTE:**

When a new material is created, the display color of the material can be specified using the Fill Color box in the Material Properties dialog. Any region that has a material assigned to it will display the corresponding material fill color.

5. On the *Conductivity* tab. Select *Data* from the Thermal Conductivity Option drop-down and press the *Data* button,
6. Enter the data points as provided at the beginning of this Tutorial,
7. Click the Volumetric Heat Capacity tab,
8. Select the Constant radio button,
9. Click the *Same Value of Unfrozen and Frozen HC* box,
10. Enter 1.95e6 for the Frozen VHC and Unfrozen VHC text field,
11. Click *OK* to accept the changes made and return to the *Materials Manager*, and
12. Repeat these steps to create the second material,
13. Click *OK* to close the *Materials Manager*,

#### **NOTE:**

To view the thermal conductivity curve of the data press the *Graph* button on the bottom right of the *Thermal Conductivity Data* dialog.

The materials will also need to be applied to the model regions. Follow these instructions in order to assign the materials to regions:



1. Open the *Regions* dialog by selecting *Model > Geometry > Regions* from the menu,
2. For the Slope region, select *2D Tutorial 1* from the Material drop-down, and
3. For the Seam region, select *2D Tutorial 2* from the Material drop-down,
14. Click *OK* to close the Regions dialog.

### e. Specify Model Output

Flux sections can be used to show the rate of heat flow across a portion of the model for a steady state analysis and the rate of heat flow moving across a portion of the model in a transient analysis.

1. Select *Draw > Flux Section* from the menu to open the Flux Section,
2. Click on the point (21,20) with the mouse,
3. To finish the Flux Section, double-click on *point (35,20)*. A blue line with an arrow on the end will be drawn across the geometry,
4. Default plots will be generated for reporting the results from the flux section, and
5. Notice that the flux section label is partially on the region boundary in the workspace. To move the label location, select the textbox in the workspace and drag it to the desired location.

#### NOTE:

Flux Section labels can be formatted in the same manner as regular text boxes.

### PLOT MANAGER (*Model > Reporting > Plot Manager*)

The *Plot Manager* dialog, when first opened will display the default plots. There are numerous plot types that can be specified to visualize the results of the model. For this tutorial an extra zoomed plot of temperature contours will be added.

1. Open the *Plot Manager* dialog by selecting *Model > Reporting > Plot Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each plot type. Click on the *Contour* button to begin adding the first contour plot. The *Plot Properties* dialog will open,
3. Enter the title *Te Zoom*,
4. Select "Te" as the variable to plot from the drop-down,
5. Move to the Zoom tab,
6. Press the Select Zoom button and select an area around the roadway or enter an origin of (19,16), width of 18, and height of 8,
7. Click *OK* to close the dialog and add the plot to the list,
8. Click *OK* to close the *Plot Manager* and return to the workspace.

**OUTPUT MANAGER (Model > Reporting > Output Manager)**

Two output files will be generated for this tutorial example: a file of temperatures to be used as initial conditions in a subsequent transient analysis, and a .dat file to view the results in ACUMESH. The file record called AcuMeshInput.dat is already present by default.

1. Open the *Output Manager* dialog by selecting *Model > Reporting > Output Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each output file type. Create the temperature file by pressed the button with the *SVHEAT* icon,
3. Click OK to close the Output Manager and return to the workspace.

**f. Run Model (Solve > Analyze)**

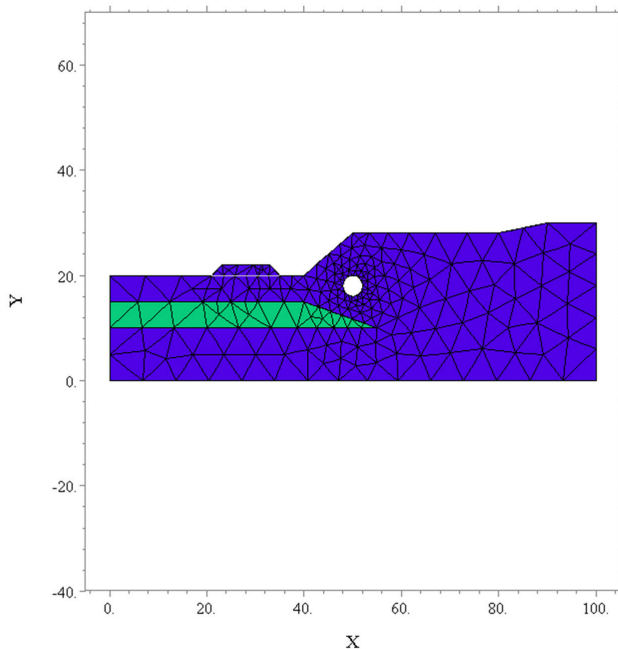
The next step is to analyze the model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model.

**g. Visualize Results (Window > AcuMesh)**

The visual results for the current model may be examined by selecting the *Window > ACUMESH* menu option.

## 7.2 Results and Discussion

After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the SVHEAT solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots.



The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas such as around the pipe contact where there is a significant change in temperature.

The temperature contours indicate that the temperature of the pipe does not significantly influence the base of roadway. Instead the heat flux due to the warming of the roadway surface has an influence of a few degrees.

Flow Vectors show both the direction and the magnitude of the heat flow at specific points in the model. Vectors show the heat flow is away from the pipeline, as anticipated.

The Flux through the base of the roadway is displayed in the report dialog showing a breakdown of the X, Y, and normal components of flow through the model. The normal flow is 16550.75 J/hr.

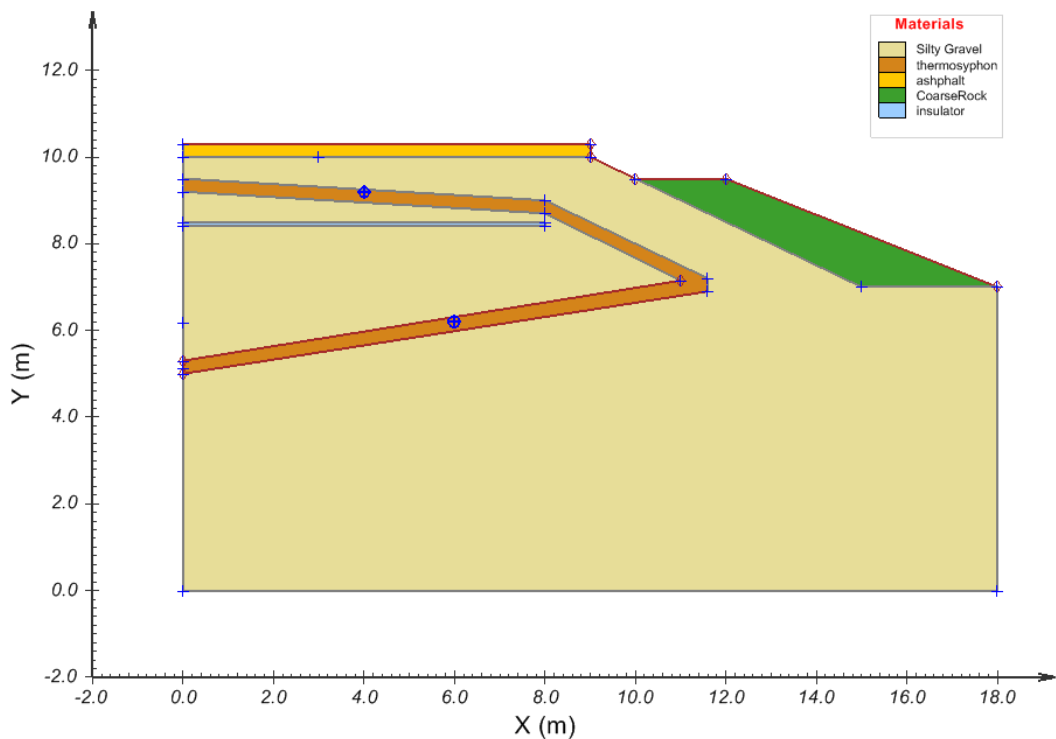
## 8 2D Hairpin Thermosyphon

The following example demonstrates how to setup and analyze a transient state two dimensional model with a thermosyphon boundary condition. The thermosyphon is used to lower the temperature of the surrounding soil. In this example the thermosyphon boundary condition is applied to a region in the geometry.

Project: Thermosyphon  
Model: HairpinThermosyphon  
System: 2D  
Type: Transient  
Minimum authorization required: Full

### Model Geometry and Description

The model geometry is composed of five regions each with a different material property. The initial soil temperature of all regions is set to 15 °C. The model surface temperature is described by a sine wave function of time. A thermosyphon boundary condition is applied to the bottom half of the thermosyphon region. The effect of the thermosyphon boundary condition on the soil temperature of the entire model will be monitored over a period of 1 year.



## 8.1 Model Setup

The following steps will be required to set up this model:

- a. Create model
- b. Enter geometry
- c. Specify boundary conditions
- d. Specify initial conditions
- e. Apply material properties
- f. Specify model output
- g. Run model
- h. Visualize results

### NOTE:

Any values on the dialogs there are not specifically mentioned in the steps below are assumed to be the default values currently present.

### a. Create Model

This tutorial assumes that you are already familiar with creating Projects and Models in SVOFFICE. If the "UserTutorial" project does not exist, click the *Create New Project* button located above the list of projects and then enter "UserTutorial" as a new project name.

Since FULL authorization is required for this tutorial, follow these steps to ensure full authorization is activated.

To begin creating the model in this tutorial create a new model in SVHEAT through the following steps:

1. Select a Project under which to organize the tutorial,
2. Press the *Create New SVHeat Model* button located above the list of models,
3. Enter **HairpinThermosyphon** in the Model Name text box,
4. Select the following entries:

Module:	SVHEAT
System:	2D
Type:	Transient
Units:	Metric
Time Units:	Days (day)
5. Click on the *World Coordinate System* tab and enter the values below,

x min = -2	x max = 18
y min = -2	y max = 12
6. Move to the *Time* tab and enter the following values for time:

Start Time:	0
Initial Increment:	0.125
Maximum Increment:	1

End Time: 365

7. Click the *OK* button to save the model and close the *New Model* dialog,
8. The new model will be automatically added to the models list and the new model will be opened.
9. On the *Display Options* dialog, set a *Horizontal and Vertical Spacing* of **0.1 m** and then click *OK* to accept the settings.

## b. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions and can be either drawn by the user or defined as a set of coordinates. Model geometry can also be imported from either .DXF files or from existing models.

This model consists of five regions which will be named Silty Gravel, Thermosyphon, Asphalt, Coarse Rock, and Insulator. These names are chosen to describe the material that will be applied to each region in the "Apply Material Properties" step below. To add the regions follow these steps:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions...* from the menu,
2. Change the region name from *R1* to *Silty Gravel*. To do this, highlight the name and type the text,
3. Press the *New* button to add a second region *R2*,
4. Change the region name from *R2* to *Thermosyphon*,
5. Repeat these steps for three remaining regions.

The shapes that define each material region will now be created. The steps to create the *Silty Gravel* region are as follows:

1. Click on the *Silty Gravel* region item in the region list box and press the *Properties...* button,
2. Click on the *New Polygon...* button to open the *New Polygon Shape* dialog,
3. Copy and paste the region coordinates at the end of this tutorial for the *Silty Gravel* region into the dialog using the *Paste Points* button (do not copy the header row),
4. Click *OK* to close the *New Polygon Shape* dialog,
5. Repeat this process to define the remaining regions,
6. Click *OK* to close the *Regions* dialog.

If all model regions have been entered correctly the geometry should look like the diagram at the beginning of this tutorial.

## c. Specify Boundary Conditions (Model > Boundaries)

Now that the model geometry has been defined, the next step is to specify the boundary conditions. Two separate climate boundary conditions are used in this model. One is used to represent the thermosyphon and the other is applied to the ground surface. The steps for

specifying these boundary conditions are as follows:

1. Open the *Climate Manager* dialog by selecting *Model > Boundaries > Climate Manager...* from the menu,

- **Define Thermosyphon Boundary Condition**

2. Click the *New...* button to open the *New Climate Data* dialog,
3. Enter *Thermosyphon* as the climate dataset name,
4. Click *OK* and the *Climate Properties* dialog will automatically appear,
5. On the *General* tab, set the *Surface Temperature Option* to **Thermosyphon** using the drop down,
6. Move to the *Thermosyphon* tab and enter the following values (note that the Evaporator size fields are left blank),  
Maximum Air Temperature: 30 °C  
Minimum Tair/Te Difference: 1 °C  
Performance Options: Constant  
P: 432,000 J/day/°C
7. Move to the *Air Temperature* tab,
8. Select the **Expression** *Air Temperature* option from the drop down,
9. In the *Constant/Expression* text box copy and paste the following function,  
$$-1 + 12 * \sin(2 * 3.141596 / 365 * t + 3.141596 / 2)$$
10. Click *OK* to close the *Climate Properties* dialog,

- **Define Natural Ground Temperature Boundary Condition**

11. Click the *New...* button to open the *New Climate Data* dialog,
12. Enter *Natural\_Ground\_Temp* as the climate dataset name,
13. Click *OK* and the *Climate Properties* dialog will automatically appear,
14. On the *General* tab, set the *Surface Temperature Option* to **Empirical with N-Factor** using the drop down,
15. Move to the *N-Factor* tab and set the *N-Factor Option* to **Constant** using the drop down,
16. Enter an *N-Factor Constant* value of **1** in the text box,
17. Move to the *Air Temperature* tab,
18. Select the **Expression** *Air Temperature* option from the drop down,
19. In the *Constant/Expression* text box copy and paste the following function,  
$$-1 + 12 * \sin(2 * 3.141596 / 365 * t + 3.141596 / 2)$$
20. Click *OK* to close the *Climate Properties* and *Climate Manager* dialogs.

The next step is to apply the boundary conditions to region boundaries in the model. The

Boundary Conditions dialog applies to the region that is currently selected in the region selector.

**NOTE:**

A region may be selected in one of the following 3 ways:

1. click on the region with the mouse cursor in the workspace
2. selecting the region in the region selector located above the workspace
3. by selecting the region row in the Regions dialog.

- **Silty Gravel Boundary Conditions**

1. Select the *Silty Gravel* region,
2. Open the *Boundary Conditions* dialog by selecting *Model > Boundaries > Boundary Conditions...* from the menu,
3. Select the point (10,9.5) from the list,
4. From the Boundary Condition drop-down select a **Climate** boundary condition,
5. Set the *Climate Name* to **Natural\_Ground\_Temp** using the drop down,
6. Click *OK* to save the Boundary Conditions and return to the workspace.

X (m)	Y (m)	Boundary Condition
0	0	No BC
18	0	Continue
18	7	Continue
15	7	Continue
10	9.5	Climate: Natural_Ground_Temp
9	10	No BC
3	10	Continue
0	10	Continue
0	6.188	Continue

- **Thermosyphon Boundary Conditions**

7. Select the *Thermosyphon* region,
8. Open the Boundary Conditions dialog by selecting *Model > Boundaries > Boundary Conditions...* from the menu,
9. Assign the boundary conditions shown in the following table,
10. Click *OK* to save the Boundary Conditions and return to the workspace.



X (m)	Y (m)	Boundary Condition
0	5	Climate: Thermosyphon
11.6	6.9	Zero Flux
11.6	7.2	Continue
8	9	No BC
0	9.5	No BC
0	9.2	No BC
8	8.7	Zero Flux
11	7.15	Climate: Thermosyphon
0	5.3	Continue

**NOTE:**

The difference between the No BC boundary condition type and the Zero Flux boundary condition type is one of placement. These two boundary condition types are interchangeable on exterior boundaries of a model and result in zero flow across the boundary. However, on interior boundaries of a model the No BC type is the default boundary condition and results in no effect on the flux across a boundary. The Zero Flow boundary condition type when applied to an internal boundary of a model results in zero flux across that boundary, i.e., a Zero Flux type is effectively a barrier that prevents any heat flow from crossing the boundary in the case of SVHeat.

- **Asphalt Boundary Conditions**

11. Select the *Asphalt* region,
12. Open the *Boundary Conditions* dialog by selecting *Model > Boundaries > Boundary Conditions...* from the menu,
13. Select the point (9,10) from the list,
14. From the Boundary Condition drop-down select a **Climate** boundary condition,
15. Set the *Climate Name* to **Natural\_Ground\_Temp** using the drop down,
16. Select the point (9,10.3) from the list and set the **Continue** boundary condition,
17. Click *OK* to save the Boundary Conditions and return to the workspace.

X (m)	Y (m)	Boundary Condition
0	10	No BC
9	10	Climate: Natural_Ground_Temp
9	10.3	Continue
0	10.3	No BC

- **Coarse Rock Boundary Conditions**

18. Select the *Coarse Rock* region,
19. Open the *Boundary Conditions* dialog by selecting *Model > Boundaries > Boundary Conditions...* from the menu,

20. Select the point (18,7) from the list,
21. From the Boundary Condition drop-down select a **Climate** boundary condition,
22. Set the *Climate Name* to **Natural\_Ground\_Temp** using the drop down,
23. Select the point (12,9.5) from the list and set the **Continue** boundary condition,
24. Click *OK* to save the Boundary Conditions and return to the workspace.

X (m)	Y (m)	Boundary Condition
18	7	Climate: Natural_Ground_Temp
12	9.5	Continue
10	9.5	No BC
15	7	Continue

#### d. Specify Initial Conditions (Model > Initial Conditions)

Initial conditions must be specified prior to solving a transient model. In this case we will specify an initial temperature for the entire model.

1. Select *Model > Initial Conditions > Settings...* from the menu,
2. Select the **Constant** option,
3. Enter a temperature of **15 °C**,
4. Click *OK* to close the dialog.

#### e. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties. Each region in the model will be assigned a different material. Define the first material called *Silty\_Gravel\_Material* as follows:

1. Open the *Materials Manager* dialog by selecting *Model > Materials > Manager...* from the menu,
2. Click the *New...* button to create a material,
3. Enter *Silty\_Gravel\_Material* for the material name,
4. Click *OK* and the *Material Properties* dialog will appear,
5. Enter the material property values given in the table below.
6. Repeat these steps for the remaining four materials.
7. Click *OK* to close the *Materials Manager* dialog.

#### Material: Silty Gravel

<b>Conductivity</b>	Thermal Conductivity Option	Constant
	Unfrozen material	75,168 J/day-m-°C
	Frozen material	75,168 J/day-m-°C
<b>Volumetric Heat Capacity</b>	Volumetric Heat Capacity Formulation	Jame-Newman
	Soil Dry Density	1,250 kg/m <sup>3</sup>
	Specific Heat Capacity of Solid Component	750 J/kg-°C
<b>SFCC</b>	From (Tef)	-0.01 °C
	To(Tep)	-0.85 °C
	SFCC Method	Expression
	SFCC	if $T_e > T_{ef}$ then $vwc$ else if $T_e > T_{ep}$ then $vwc / (T_{ef} - T_{ep}) * (T_e - T_{ep}) + 0.02$ else 0.02
	m2i	if $T_e < T_{ef}$ and $T_e > T_{ep}$ then $vwc / (T_{ef} - T_{ep})$ else 0
<b>VWC</b>	SatVWC	0.4
	VWC	0.4

**Material: Thermosyphon**

<b>Conductivity</b>	Thermal Conductivity Option	Constant
	Unfrozen material	3.456E+07 J/day-m-°C
	Frozen material	3.456E+07 J/day-m-°C
<b>Volumetric Heat Capacity</b>	Volumetric Heat Capacity Formulation	Constant
	Unfrozen	5,000 J/m <sup>3</sup> -°C
	Frozen	5,000 J/m <sup>3</sup> -°C
<b>SFCC</b>	From (Tef)	-0.01 °C
	To(Tep)	-0.5 °C
	SFCC Method	None
<b>VWC</b>	SatVWC	0.35
	VWC	0.001

**Material: Asphalt**

<b>Conductivity</b>	Thermal Conductivity Option	Constant
	Unfrozen material	64,800 J/day-m-°C
	Frozen material	64,800 J/day-m-°C
<b>Volumetric Heat Capacity</b>	Volumetric Heat Capacity Formulation	Constant
	Unfrozen	2,520,000 J/m <sup>3</sup> -°C
	Frozen	2,520,000 J/m <sup>3</sup> -°C
<b>SFCC</b>	From (Tef)	-0.01 °C
	To(Tep)	-0.5 °C
	SFCC Method	None
<b>VWC</b>	SatVWC	0.35
	VWC	0.01

**Material: CoarseRock**

<b>Conductivity</b>	Thermal Conductivity Option	Constant
	Unfrozen material	43,200 J/day-m-°C
	Frozen material	43,200 J/day-m-°C
<b>Volumetric Heat Capacity</b>	Volumetric Heat Capacity Formulation	Jame Newman
	Soil Dry Density	1700 kg/m <sup>3</sup>
	Specific Heat Capacity of Solid Component	800 J/kg-°C
<b>SFCC</b>	From (Tef)	-0.01 °C
	To(Tep)	-0.5 °C
	SFCC Method	None
<b>VWC</b>	SatVWC	0.35
	VWC	0.01

**Material: Insulator**

<b>Conductivity</b>	Thermal Conductivity Option	Constant
	Unfrozen material	2,419.2 J/day-m-°C
	Frozen material	2,419.2 J/day-m-°C
<b>Volumetric Heat Capacity</b>	Volumetric Heat Capacity Formulation	Constant
	Unfrozen	125,600 J/m <sup>3</sup> -°C
	Frozen	125,600 J/m <sup>3</sup> -°C
<b>SFCC</b>	From (Tef)	-0.01 °C
	To(Tep)	-0.5 °C
	SFCC Method	None
<b>VWC</b>	SatVWC	0.35
	VWC	0.35

The materials will need to be applied to the model regions by following these steps:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions...* from the menu,
2. For the *Silty Gravel* region, select *Silty\_Gravel\_Material* from the Material drop down list,
3. Repeat this step for each of the remaining regions and corresponding materials,
4. Click *OK* to close the *Regions* dialog.

#### f. Specify Model Output (Model > Reporting)

In this model the default plots will be created. In addition, a group of two *History* plots will be created separately in order to view the Temperature profiles at two locations in the *Thermosyphon* region over time. The history plot locations may be seen in the Model Geometry section as the blue circles in the Thermosyphon region.

To set up these plots follow the steps below.

1. Open the *Plot Manager* dialog by selecting *Model > Reporting > Plot Manager...* from the menu,
2. Move to the *Point* tab,
3. Click the *Multiple Entry...* button,
4. Set the *Variable* to **Temperature** by selecting the value from the drop down list,
5. Set the *Plot Type* to **History** by selecting the value from the drop down list,
6. In the *Group* text box type **Soil Temperature**,
7. Type or copy the data from the table below (do not include the header row) and click the *Paste* button to paste the data into the dialog,
8. Click *OK* to close the *Multiple Entry* and *Plot Manager* dialogs.

X (m)	Y (m)	Title
6	6.2	Temp at evaporator
4	9.2	Temp at condenser

#### g. Run Model (Solve > Analyze)

The next step is to analyze the model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model. After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the FlexPDE solver. Right-click the mouse and select "Maximize", or double-click, to enlarge any of the thumbnail plots.

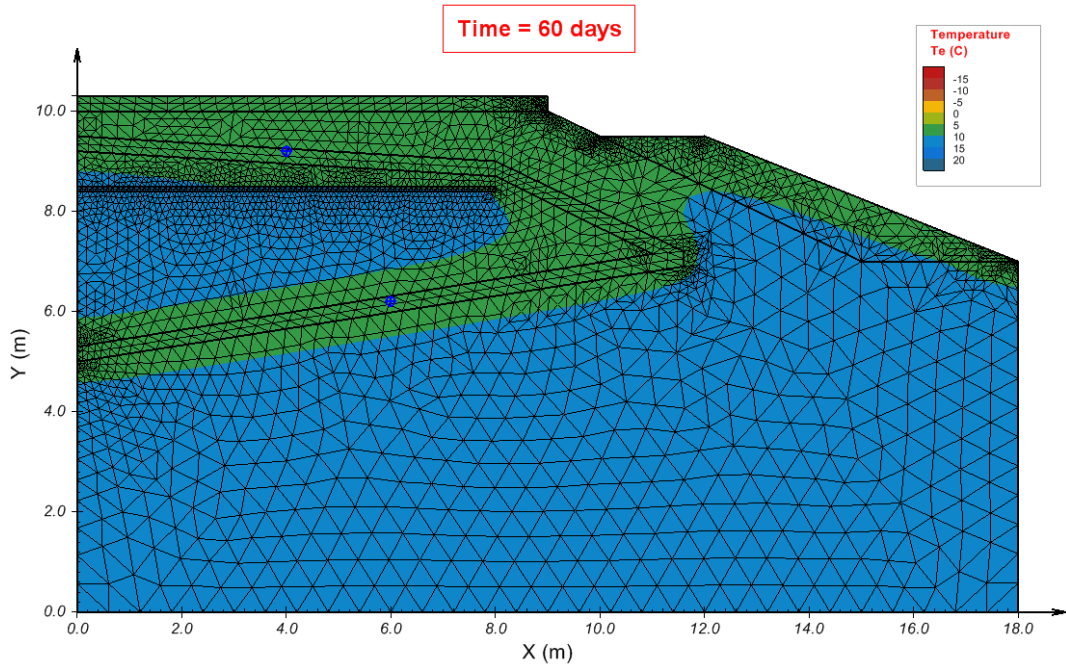
These reports are intended to provide the user with low-quality graphs which give a rough indication of the results. Creating professional-quality visualizations of the results can be accomplished with ACUMESH software.

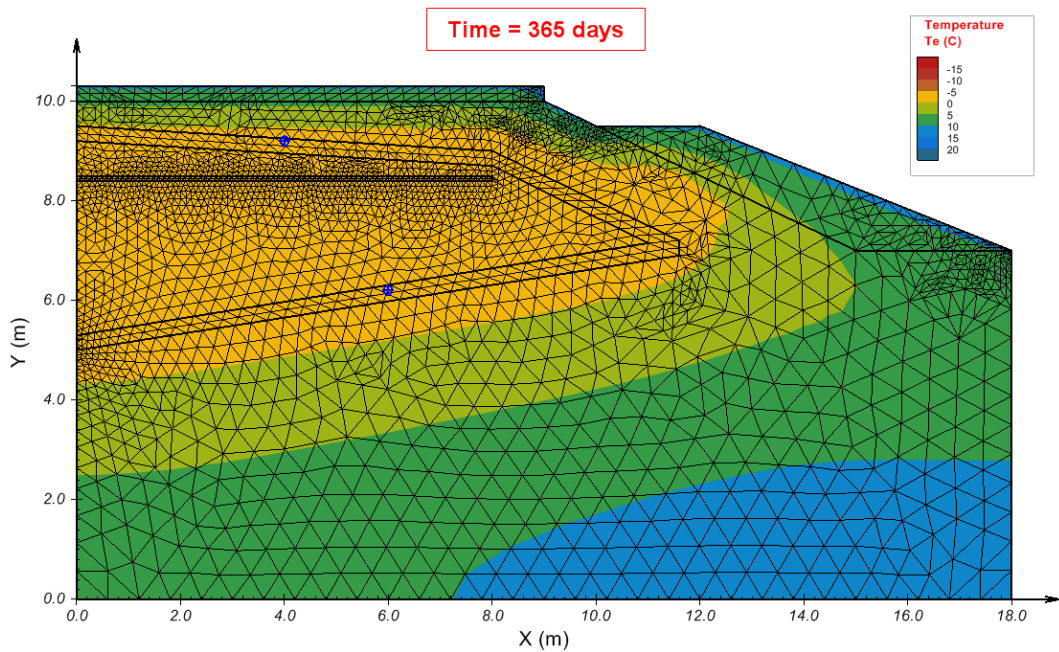
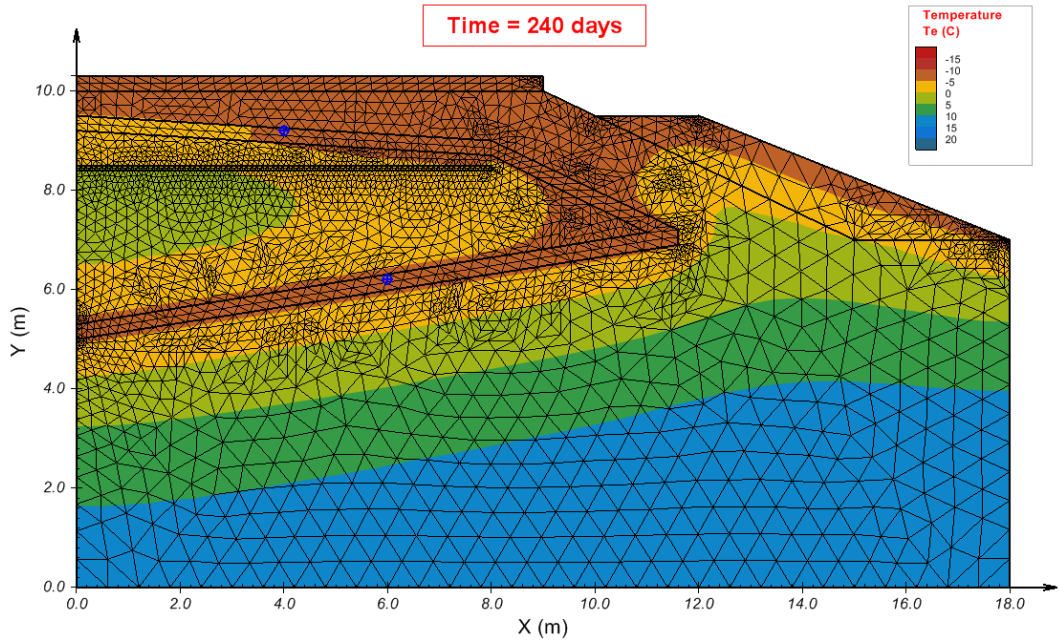
#### h. Visualize Results (Window > AcuMesh)

The visual results for the current model may be examined by selecting the *Window > ACUMESH* menu option.

## 8.2 Results and Discussion

The default plot that appears in Acumesh is a contour plot of the Temperature variable. The finite element mesh used to solve the model is also displayed by default. The selected time step may be changed by using the drop down in the workspace toolbar. The screenshots below show the progression of the cooling of the soil starting from the thermosyphon and moving outward at the time steps 60 days, 240 days, and 365 days.



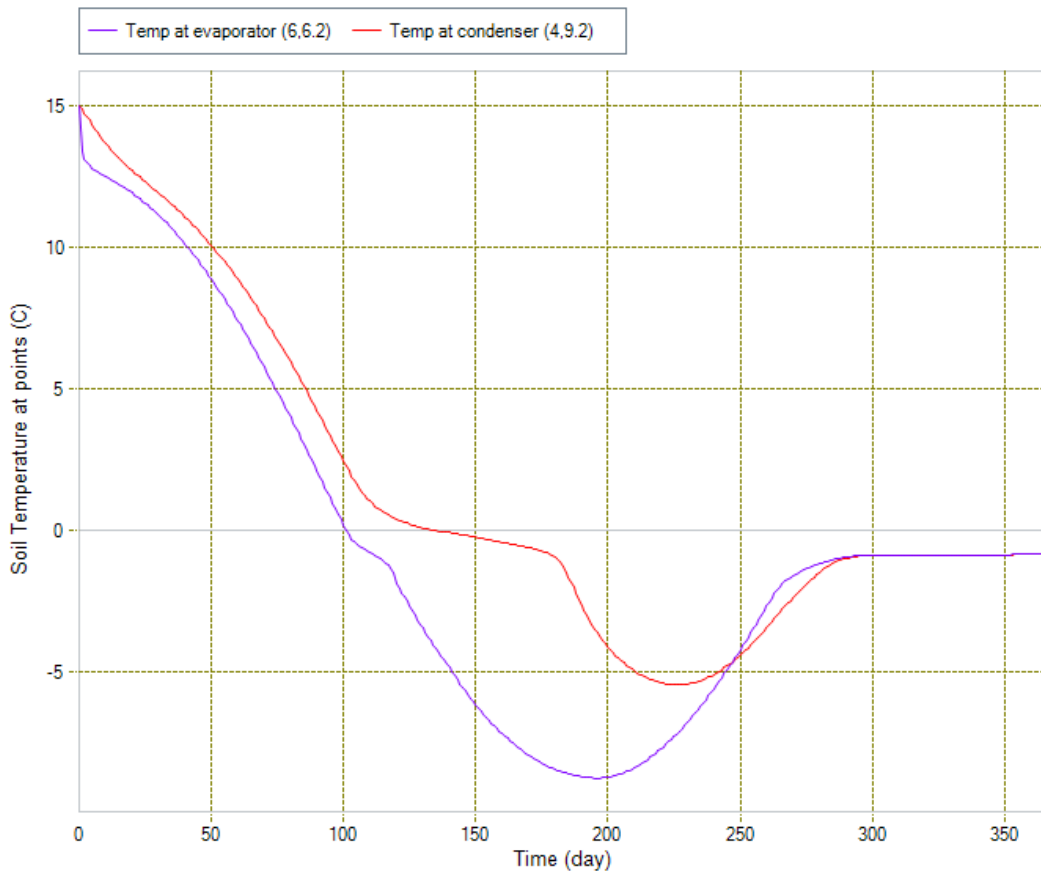


The history plots defined in the Model Output section provide a summary of the soil temperature at two points in the Thermosyphon region over time. This plot may be viewed in

Acumesh by following these steps:

1. Select *Graphs > Plot Manager...* menu item,
2. Move to the *Point* tab,
3. Select the *Soil Temperature* plot from the list of plots and click the *Graph* button.

The results shown in the plot illustrate the soil temperature changes over time at the two history points. The effects of the thermosyphon boundary condition and the natural ground temperature boundary condition on the soil temperature at the two history points can be seen in the plot. Because the thermosyphon boundary condition was not applied to the thermosyphon region near the history point (4,9.2) there is a delay in the effect of the thermosyphon boundary condition at the point (6,6.2). This delay causes the temperature at the history point (4,9.2) to remain higher than at the point (6,6.2) until roughly day 300 when the temperature begins to equalize between the two points.





## 8.3 Model Data

### Geometry

#### Region: Silty Gravel

X (m)	Y (m)
0	0
18	0
18	7
15	7
10	9.5
9	10
3	10
0	10
0	6.188

#### Region: Thermosyphon

X (m)	Y (m)
0	5
11.6	6.9
11.6	7.2
8	9
0	9.5
0	9.2
8	8.7
11	7.15
0	5.3

#### Region: Asphalt

X (m)	Y (m)
0	10
9	10
9	10.3
0	10.3

#### Region: Coarse Rock

X (m)	Y (m)
18	7
12	9.5
10	9.5
15	7

#### Region: Insulator

X (m)	Y (m)
0	8.4
8	8.4
8	8.5
0	8.5

[Return to Enter Geometry Section](#)

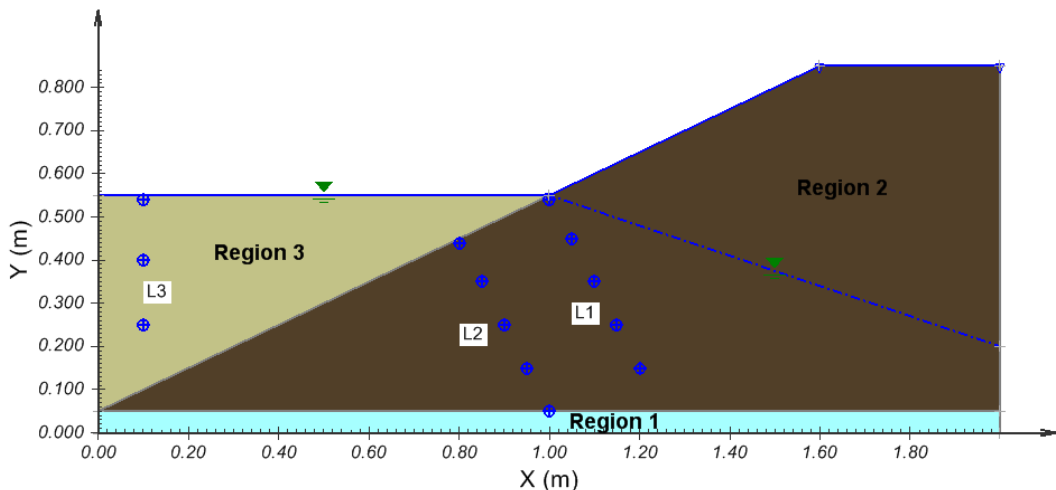
## 9 2D Canal Bank Freezing and Thawing

This example illustrates hydrothermal coupling (SVHeat and SVFlux) in the simulation of soil and ice freeze-thaw behavior on a canal bank. The water in the canal is also included in the analysis. The model simulation time is 50 hours. The canal bank is freezing during the first 25 hours after which time the thawing process occurs for the remainder of the simulation time.

Project: GeoThermal  
Model: CanalBankFreezingThawing\_50h  
System: 2D  
Type: Transient  
Minimum authorization required: Full

### Model Geometry and Description

The model geometry consists of 3 regions. Region 1 is a layer of sand 0.6 m in thickness. Region 2 is the canal bank. It is 2 m wide and 0.85 m high with slope of 1h:2w. Region 3 is used to represent the water. The water itself is usually not included in seepage modeling, but in this example it is utilized to simulate the canal ice thickness during freeze-thaw. The blue points in the model geometry below are used to record the soil/ice temperature at specific locations.



### 9.1 Model Setup

The following steps will be required to set up this model:

- Create model
- Enter geometry
- Specify SVHeat initial conditions
- Specify SVHeat boundary conditions
- Combine SVFlux with SVHeat
- Apply SVHeat material properties
- Specify SVHeat model output

- h. Specify SVFlux boundary conditions
- i. Specify SVFlux initial conditions
- j. Apply SVFlux material properties
- k. Run coupled SVHeat and SVFlux model
- l. Visualize results

**NOTE:**

Any values on the dialogs there are not specifically mentioned in the steps below are assumed to be the default values currently present.

**a. Create Model**

This tutorial assumes that you are already familiar with creating Projects and Models in SVOFFICE. If the "UserTutorial" project does not exist, click the *Create New Project* button located above the list of projects and then enter "UserTutorial" as a new project name.

Since FULL authorization is required for this tutorial, follow these steps to ensure full authorization is activated.

To begin creating the model in this tutorial create a new model in SVHEAT through the following steps:

1. Select a Project under which to organize the tutorial,
2. Press the *Create New SVHeat Model* button located above the list of models,
3. Enter **SVHeat\_CanalBank** in the Model Name text box,
4. Select the following entries:
 

Module:	SVHEAT
System:	2D
Type:	Transient
Units:	Metric
Time Units:	hr
5. Click on the *World Coordinate System* tab and enter the values below,
 

x min = 0	x max = 2.1
y min = 0	y max = 1
6. Move to the *Time* tab and enter the following values for time:
 

Start Time:	0
Initial Increment:	0.01
Maximum Increment:	0.1
End Time:	50
7. Click the *OK* button to save the model and close the *New Model* dialog,
8. The new model will be automatically added to the models list and the new model will be opened.
9. On the *Display Options* dialog, set a *Horizontal and Vertical Spacing* of **0.05 m** and then click *OK* to accept the settings.

## b. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions and can be either drawn by the user or defined as a set of coordinates. Model geometry can also be imported from either .DXF files or from existing models.

This model consists of three regions which are named *Filter*, *CanalBank*, and *Water*. To add the regions follow these steps:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions...* from the menu,
2. Change the region name from R1 to *Filter*. To do this, highlight the name and type the text,
3. Press the *New* button to add a second region *R2*,
4. Change the region name from R2 to *CanalBank*,
5. Repeat this step for the remaining region.

The shapes that define each material region will now be created. The steps to create the *Filter* region are as follows:

1. Click on the *Filter* region item in the region list box and press the *Properties...* button,
2. Click on the *New Polygon...* button to open the *New Polygon Shape* dialog,
3. Copy and paste the region coordinates from the table below into the dialog using the *Paste Points* button (do not copy the header row),
4. Click OK to close the *New Polygon Shape* dialog and *Region Properties* dialogs,
5. Repeat these steps to define the remaining region shapes,
6. Press *OK* to close the *Regions* dialog.

### Region: Filter

X (m)	Y (m)
0	0
2	0
2	0.05
0	0.05

### Region: CanalBank

X (m)	Y (m)
0	0.05
2	0.05
2	0.85
1.6	0.85
1	0.55

### Region: Water

X (m)	Y (m)
0	0.05
1	0.55
0	0.55

### c. Specify SVHeat Initial Conditions (Model > Initial Conditions)

Initial conditions must be specified prior to solving a transient model. In this case we will specify a global initial temperature.

1. Select *Model > Initial Conditions > Settings...* from the menu,
2. Select the **Constant** option,
3. Enter a temperature of **5 °C**,
4. Click *OK* to close the dialog.

### d. Specify SVHeat Boundary Conditions (Model > Boundaries)

Now that the model geometry has been defined, the next step is to specify the boundary conditions. The freezing rate for the surface of the canal bank and the water regions will be set to a temperature expression. The remaining model boundaries are set to the default value of No BC. The steps for specifying these boundary conditions are as follows:

The Boundary Conditions dialog applies to the region that is currently selected in the region selector.

#### NOTE:

A region may be selected in one of the following 3 ways:

1. click on the region with the mouse cursor in the workspace
2. selecting the region in the region selector located above the workspace
3. by selecting the region row in the Regions dialog.

#### NOTE:

The difference between the No BC boundary condition type and the Zero Flux boundary condition type is one of placement. These two boundary condition types are interchangeable on exterior boundaries of a model and result in zero flow across the boundary. However, on interior boundaries of a model the No BC type is the default boundary condition and results in no effect on the flux across a boundary. The Zero Flow boundary condition type when applied to an internal boundary of a model results in zero flux across that boundary, i.e., a Zero Flux type is effectively a barrier that prevents any heat flow from crossing the boundary in the case of SVHeat.

#### • CanalBank region boundary conditions

1. Select the *CanalBank* region,
2. Open the *Boundary Conditions* dialog by selecting *Model > Boundaries > Boundary Conditions...* from the menu,
3. Assign the boundary conditions as shown in the following table,
4. Click *OK* to save the Boundary Conditions and return to the workspace.

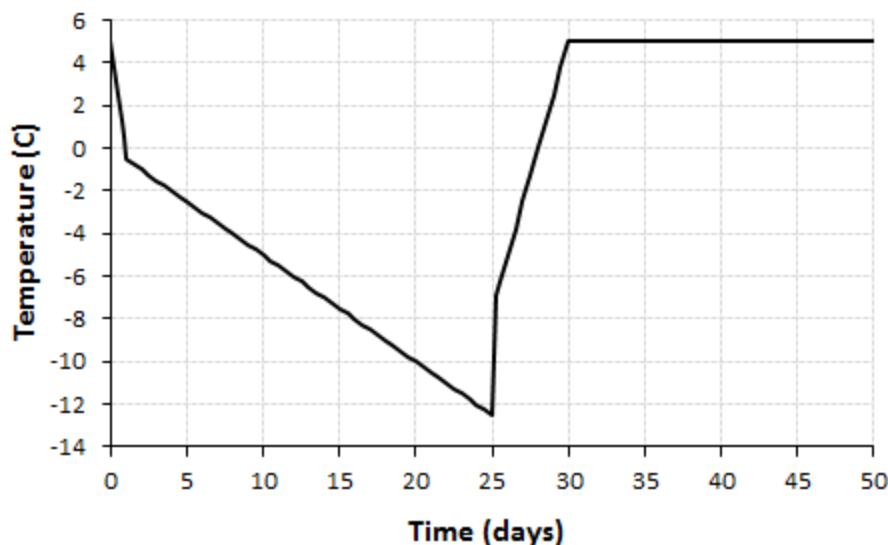
X (m)	Y (m)	Boundary Condition
0	0.05	No BC
2	0.05	No BC
2	0.85	Temperature Expression: if $t < 1$ then $5 - 5*t$ else if $t \leq 25$ then $-0.5*t$ else if $t < 30$ then $-7.5 + 2.5 *(t - 25)$ else 5
1.6	0.85	Continue
1	0.55	No BC

- **Water region boundary conditions**

5. Select the *Water* region,
6. Open the Boundary Conditions dialog by selecting Model > Boundaries > Boundary Conditions... from the menu,
7. Assign the boundary conditions as shown in the following table,
8. Click *OK* to save the Boundary Conditions and return to the workspace.

X (m)	Y (m)	Boundary Condition
0	0.05	No BC
1	0.55	Temperature Expression: if $t < 1$ then $5 - 5*t$ else if $t \leq 25$ then $-0.5*t$ else if $t < 30$ then $-7.5 + 2.5 *(t - 25)$ else 5
0	0.55	No BC

The shape of the Temperature Expression function is shown in the following figure.



### e. Combine SVFlux with SVHeat (File > Add Coupling)

Modeling of SVFlux and SVHeat can be done independently or coupled by specifying SVFlux and SVHeat components in the same model file. This methodology makes it easy to use the finite-element volumetric water content results in the thermal modeling software. This step will also enable the modeling of both the conduction and convection processes. The steps to combine SVFlux with SVHeat are as follows:

1. Press *File > Save* to save a copy of the completed steps so far in the current model because the *Add Coupling* operation creates a new model,
2. Select *File > Add Coupling...*,
3. The *Add Coupling* dialog will be displayed,
4. Check the *SVFlux* box,
5. Note that this process creates a new model file in the same Project with the combined components,
6. Enter **Coupled\_CanalBank** as the *New File Name*,
7. Click *OK* to close the dialog.

### f. Apply SVHeat Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the SVHeat component. Each region in the model will be assigned a different material. Define the first material called *Sand* as follows:

1. Open the *Materials Manager* dialog by selecting *Model > Materials > Manager...* from the menu,
2. Click the *New...* button to create a material,
3. Enter *Sand* for the material name,
4. Click *OK* and the *Material Properties* dialog will appear,
5. Enter the material property values given in the table below. Note that the SWCC Method parameters and Hydraulic Conductivity Method parameters will be entered later into the SVFlux material properties dialog.
6. Click *OK* to close the *Material Properties* dialog,
7. Repeat these steps for the remaining two materials.
8. Click *OK* to close the *Materials Manager* dialog.

#### Material: Sand



<b>Conductivity</b>	Thermal Conductivity Option	Johansen
	Material State	Crushed
	Material Type	Coarse
	Dry/Sat Conductivity	Calculate
	Solid Conductivity	Solid Conductivity
	Solid Component	20,160 J/hr-m-°C
<b>Volumetric Heat Capacity</b>	Volumetric Heat Capacity Formulation	Jame-Newman
	Solid Dry Density	1,500 kg/m <sup>3</sup>
	Specific Heat Capacity of Solid Component	700 J/kg-°C
<b>SFCC</b>	From (Tef)	-0.05 °C
	To (Tep)	-0.5 °C
	SFCC Method	Estimated by SWCC
	SWCC Method	Fredlund and Xing Fit
<b>Hydraulic Permeability Reduction</b>	Permeability Reduction Method	SWCC Method
	Hydraulic Conductivity Method	Modified Campbell Estimation

**Material: Silt**

<b>Conductivity</b>	Thermal Conductivity Option	Johansen
	Material State	Crushed
	Material Type	Fine
	Dry/Sat Conductivity	Calculate
	Solid Conductivity	Solid Conductivity
	Solid Component	29,500 J/hr-m-°C
<b>Volumetric Heat Capacity</b>	Volumetric Heat Capacity Formulation	Jame-Newman
	Solid Dry Density	1,250 kg/m <sup>3</sup>
	Specific Heat Capacity of Solid Component	900 J/kg-°C
<b>SFCC</b>	From (Tef)	-0.08 °C
	To (Tep)	-0.5 °C
	SFCC Method	Estimated by SWCC
	SWCC Method	Fredlund and Xing Fit
<b>Hydraulic Permeability Reduction</b>	Permeability Reduction Method	SWCC Method
	Hydraulic Conductivity Method	Modified Campbell Estimation

**Material: Water**

<b>Conductivity</b>	Thermal Conductivity Option	Expression: if $T_e < 0$ then 8280 else 2178
<b>Volumetric Heat Capacity</b>	Volumetric Heat Capacity Formulation	Constant
	Unfrozen Volumetric Heat Capacity	4,184,000 J/m <sup>3</sup> -°C
	Frozen Volumetric Heat Capacity	2,094,000 J/m <sup>3</sup> -°C
<b>SFCC</b>	From (Tef)	-0.01 °C
	To (Tep)	-0.5 °C
	SFCC Method	Expression
	SFCC	If $T_e > T_{ef}$ then 0.99 else 0
	m2i	2
<b>Hydraulic Permeability Reduction</b>	Permeability Reduction Method	SWCC Method
	Hydraulic Conductivity Method	Modified Campbell Estimation

The materials need to be applied to the model regions by following these steps:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions...* from the menu,
2. For the *Filter* region, select *Sand* from the Material drop down list,
3. For the *CanalBank*, select *Silt* from the Material drop down list,
4. For the *Water* region, select *Water* from the Material drop down list,
5. Click *OK* to close the *Regions* dialog.

### g. Specify SVHeat Model Output (Model > Reporting)

Two types of output may be specified:

- i) Plot Manager: Output displayed during model solution
- ii) Output Manager: Standard finite element files written to disk for visualization in Acumesh or for inputting to other finite element packages

In addition to the default *Output Manager* plots a contour plot for the *Ice Content* will be created. In addition to the default *Plot Manager* plots, three groups of *History* plots will also be created in order to view the *Temperature* profiles over time along three lines. The history plot locations may be seen in the Model Geometry section as the blue circles along the lines labeled *L1*, *L2* and *L3*.

To set up these plots follow the steps below.

#### • PLOT MANAGER (Model > Reporting > Plot Manager)

1. Open the *Plot Manager* dialog by selecting *Model > Reporting > Plot Manager...* from the menu,
2. Move to the *Point* tab,
3. Click the *Multiple Entry...* button,
4. Set the *Variable* to **Temperature** by selecting the value from the drop down

- list,
- Set the *Plot Type* to **History** by selecting the value from the drop down list,
  - In the *Group* text box type **Soil Temp at L1**,
  - Copy the data from the table below (do not include the header row) and click the *Paste* button to paste the data into the dialog,
  - Click *OK* to close the *Multiple Entry* dialog,
  - Repeat these steps for the two remaining history plot groups,
  - Click *OK* to close the *Plot Manager* dialog.

**History Plot Group: Soil Temp at L1**

X (m)	Y (m)	Title
1	0.54	L1PT1
1.05	0.45	L1PT2
1.1	0.35	L1PT3
1.15	0.25	L1PT4
1.2	0.15	L1PT5

**History Plot Group: Soil Temp at L2**

X (m)	Y (m)	Title
0.8	0.44	L2PT1
0.85	0.35	L2PT2
0.9	0.25	L2PT3
0.95	0.15	L2PT4
1	0.05	L2PT5

**History Plot Group: Ice Temp at L3**

X (m)	Y (m)	Title
0.1	0.54	L3PT1
0.1	0.4	L3PT2
0.1	0.25	L3PT3

- OUTPUT FILES** ([Model > Reporting > Output Manager](#))

The output .dat file will be used to transfer model solution results to the Acumesh software. The default AcuMeshInput.dat file is listed in the Output Manager by default for every model. It contains the most common variables used for visualization in AcuMesh. To add the Ice Content variable to Acumesh output follow these steps:

- Open the *Output Manager* dialog by selecting *Model > Reporting > Output Manager...* from the menu,
- Select the row containing the AcuMeshInput.dat item in the list of output files and click the *Properties...* button,
- On the *Description* tab, select the row containing the **Ice Content** variable from the list of *Available Variables*,
- Click on the *Add Selected Variables to Output File Destination* button to move

the *Ice Content* variable into the *Selected Variables* list,

5. Click *OK* to close the *Output File Properties* and *Output Manager* dialogs and return to the workspace.

## h. Specify SVFlux Boundary Conditions (Model > Boundaries)

The SVFlux boundary conditions are set up to be consistent with the initial water table and consist of a head constant values. The steps for specifying these boundary conditions are as follows:

- **CanalBank region boundary conditions**

1. First switch to SVFLUX mode by selecting *Window > SVFlux* from the menu,
2. Select the *CanalBank* region,
2. Open the *Boundary Conditions* dialog by selecting *Model > Boundaries > Boundary Conditions...* from the menu,
3. Assign the boundary conditions as shown in the following table,
4. Click *OK* to save the Boundary Conditions and return to the workspace.

X (m)	Y (m)	Boundary Condition
0	0.05	No BC
2	0.05	Head Constant: 0.2 m
2	0.2	No BC
2	0.85	Continue
1.6	0.85	Continue
1	0.55	Continue

- **Water region boundary conditions**

5. Select the *Water* region,
6. Open the Boundary Conditions dialog by selecting *Model > Boundaries > Boundary Conditions...* from the menu,
7. Assign the boundary conditions as shown in the following table,
8. Click *OK* to save the Boundary Conditions and return to the workspace.

X (m)	Y (m)	Boundary Condition
0	0.05	No BC
1	0.55	Head Constant: 0.55 m
0	0.55	Continue

- **Filter region boundary conditions**

9. Select the *Filter* region,
10. Open the Boundary Conditions dialog by selecting *Model > Boundaries > Boundary Conditions...* from the menu,
11. Assign the boundary conditions as shown in the following table,
12. Click *OK* to save the Boundary Conditions and return to the workspace.

X (m)	Y (m)	Boundary Condition
0	0	No BC
2	0	Head Constant: 0.2 m
2	0.05	No BC
0	0.05	Continue

### i. Specify SVFlux Initial Conditions (Model > Initial Conditions)

The remaining steps all apply to the SVFlux software. To switch the model view to the SVFlux component of the model select *Window > SVFlux* from the menu or click on the SVFlux icon on the sidebar near the top left of the screen.

For the SVFlux initial conditions a water table will be specified.

1. Select *Model > Initial Conditions > Settings...* from the menu,
2. Select the following settings,
 

Apply one Initial Condition to: Entire Model  
 Initial Conditions for Model: Water Table  
 Suction: No maximum
3. Click *OK* to close the dialog,
4. Select *Model > Initial Conditions > Water Table...* from the menu,
5. Copy the data from the table below (do not include the header row) and click the *Paste* button to paste the data into the dialog,
6. Click *OK* to close the *Initial Water Table* dialog.

X (m)	Y (m)
0	0.55
1	0.55
2	0.2

### j. Apply SVFlux Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the SVFlux component. The materials were created and applied to the model regions as part of the SVHeat component. The only steps left to perform in SVFlux are to assign the SVFlux specific material properties. To assign the properties follow these steps:

1. Open the *Materials Manager* dialog by selecting *Model > Materials > Manager...* from the menu,
2. Select the *Sand* material in the list click the *Properties...* button,

3. Enter the material property values for the *Sand* material given in the table below,
4. Click *OK* to close the *Material Properties* dialog,
5. Repeat these steps for the the *Silt* material.
6. Select the *Water* material in the list click the *Change Type...* button,
7. The New Data Type will be *Saturated* Click *OK* to close the *Change Type* dialog,
8. Enter the material property values for the *Water* material given in the table below,
4. Click *OK* to close the *Material Properties* dialog,
6. Click *OK* to close the *Materials Manager* dialog.

**Unsaturated Material: Sand**

<b>Volumetric Water Content</b>	Saturated VWC	0.32
	SWCC	Fredlund and Xing Fit
	af	20 kPa
	nf	1
	mf	2
	hr	500 kPa
	Suction	0.1 kPa
<b>Hydraulic Conductivity</b>	ksat Options	Constant
	Constant ksat	0.036 m/hr
	Unsaturated Hydraulic Conductivity	Modified Campbell Estimation
	kmin	3.6e-7 m/hr
	MCampbell p	5

**Unsaturated Material: Silt**

<b>Volumetric Water Content</b>	Saturated VWC	0.45
	SWCC	Fredlund and Xing Fit
	af	200 kPa
	nf	1
	mf	2
	hr	1000 kPa
	Suction	0.1 kPa
<b>Hydraulic Conductivity</b>	ksat Options	Constant
	Constant ksat	0.0036 m/hr
	Unsaturated Hydraulic Conductivity	Modified Campbell Estimation
	kmin	3.6e-7 m/hr
	MCampbell p	5

**Saturated Material: Water**

<b>Hydraulic Conductivity</b>	ksat Options	Constant
	Constant ksat	1000 m/hr
<b>Volumetric Water Content</b>	Saturated VWC	0.99

### k. Run Coupled SVHeat and SVFlux Model (Solve > Analyze)

The next step is to analyze the coupled model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model. After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the FlexPDE solver. Right-click the mouse and select "Maximize", or double-click, to enlarge any of the thumbnail plots.

These reports are intended to provide the user with low-quality graphs which give a rough indication of the results. Creating professional-quality visualizations of the results can be accomplished with ACUMESH software.

#### NOTE:

If a much faster solution time is desired the FlexPDE REGRID setting for adaptive mesh refinement may be turned off for this model. In general, for a given model it is recommended that this setting be left turned on to guarantee the specified accuracy of the solution. If a faster solution time (at the expense of a less accurate solution) is the goal then the safer alternative to disabling the REGRID setting is to increase the FlexPDE error limits on the *FEM Options* dialog. For purposes of demonstrating the solution of this model, disabling the REGRID setting is acceptable. To do this follow these steps:

1. Click *Mesh > Settings...* to open the *Mesh Settings* dialog,
2. On the *Global* tab, uncheck the *REGRID* check box,
3. Click *OK* to close the dialog.

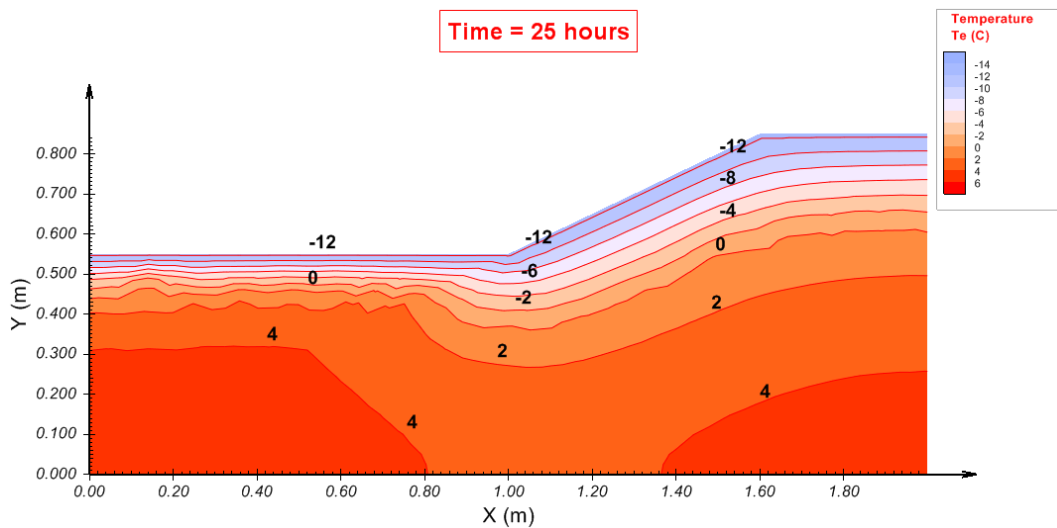
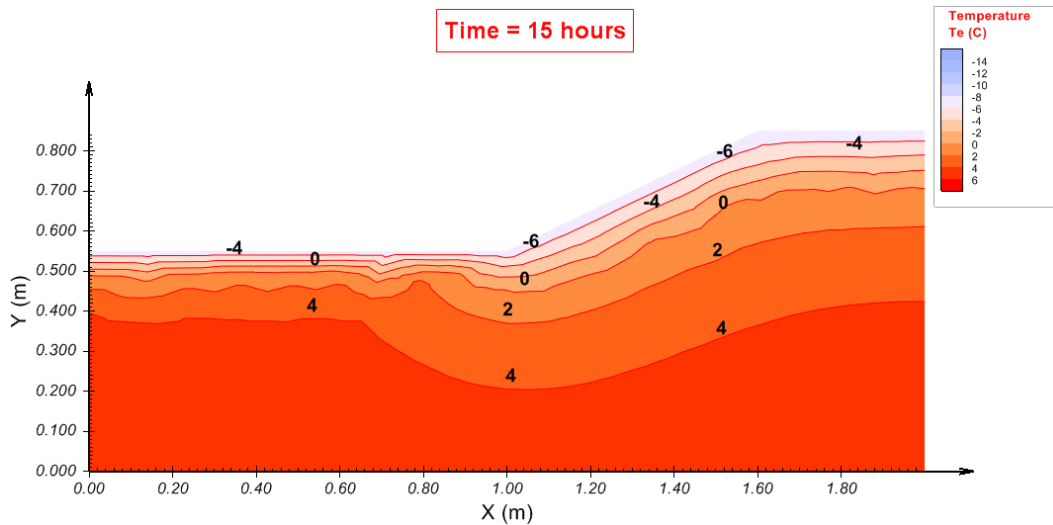
### l. Visualize Results (Window > AcuMesh)

The visual results for the current model may be examined by selecting the *Window > ACUMESH* menu option.

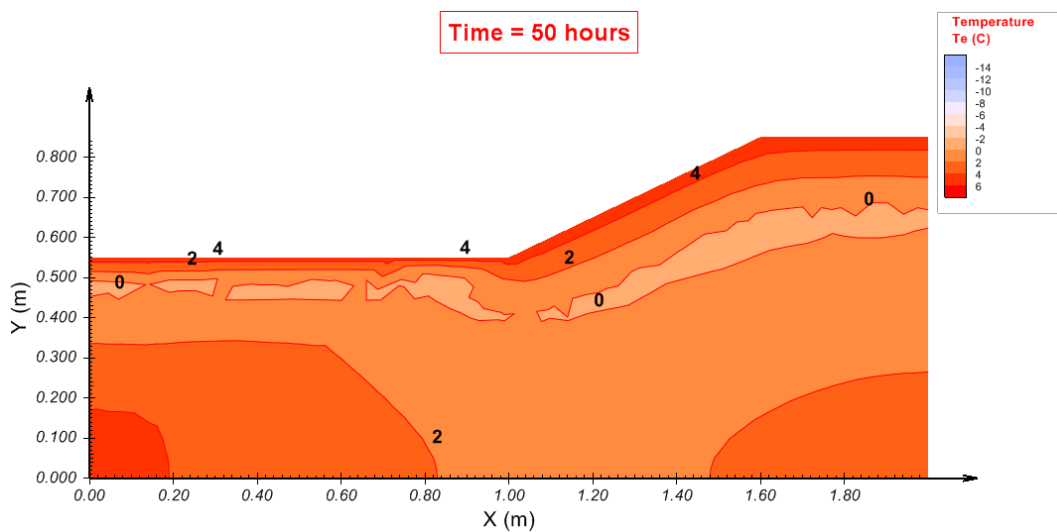
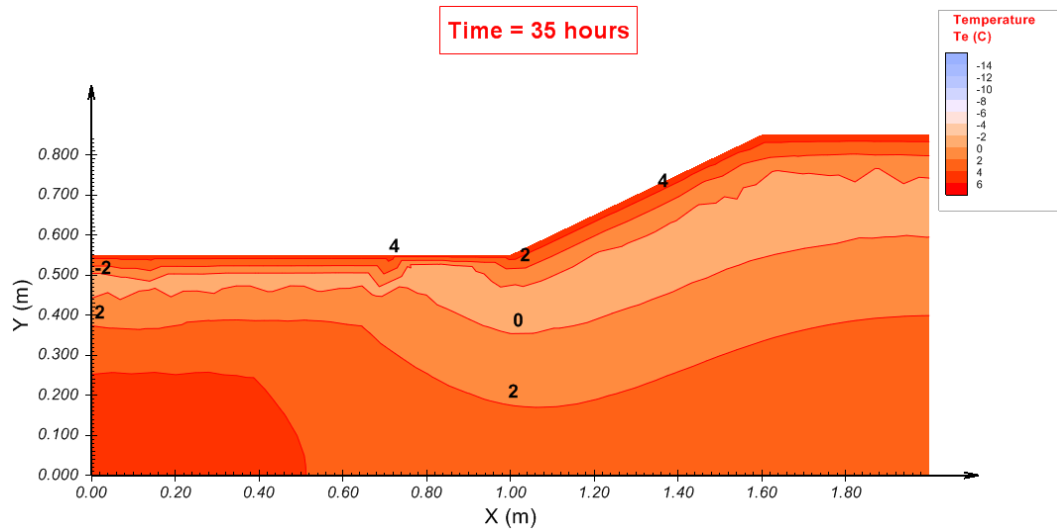
## 9.2 Results and Discussion

A view of the model solution is shown in Acumesh using the default view options. The contour plot variable may be changed by clicking *Plot > Contours...* from the menu and selecting a different variable from the *Variable* drop down on the *General* tab. The finite element mesh used to solve the model is also displayed by default. The mesh can be hidden clicking *Plot > Mesh...* and unchecking the *Draw Original Mesh* check box. The selected time step currently displayed on the screen may be changed by using the drop down in the workspace toolbar.

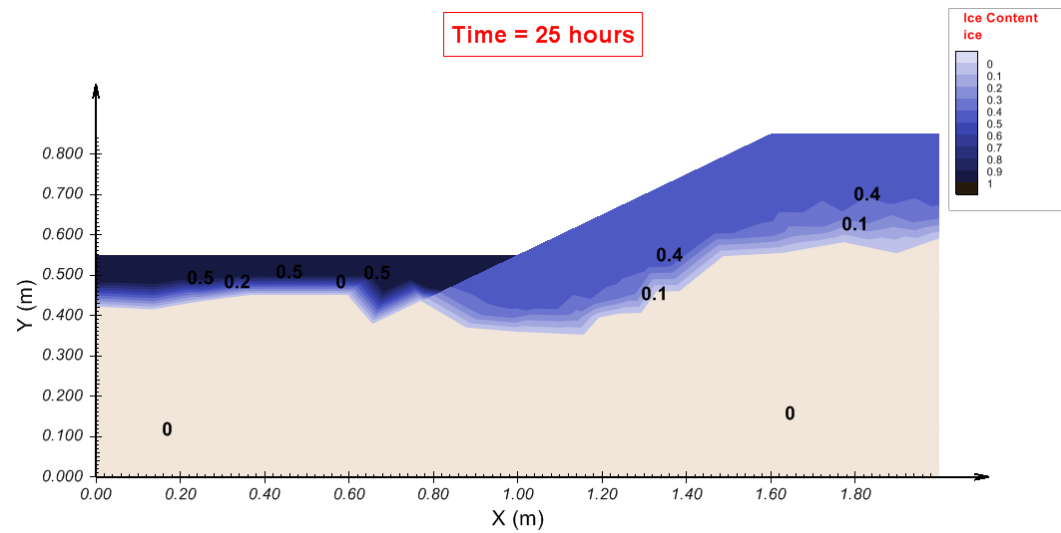
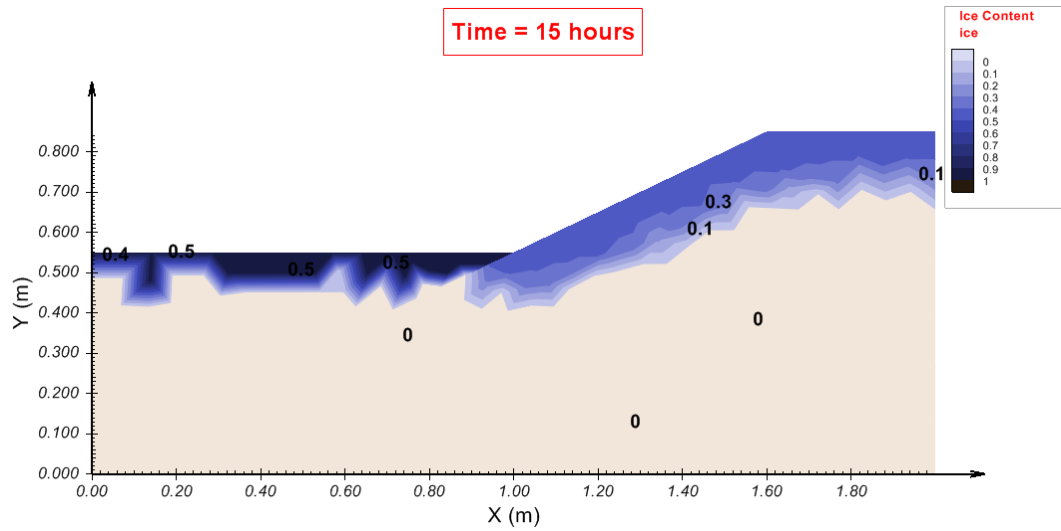
The following screenshots show the progression of the freezing and thawing of the water and canal bank. The freezing period occurs for the first 25 hours followed by a thawing period until the end of simulation time. The temperature on the canal bank decreases faster than on canal water level because of the larger latent heat release.

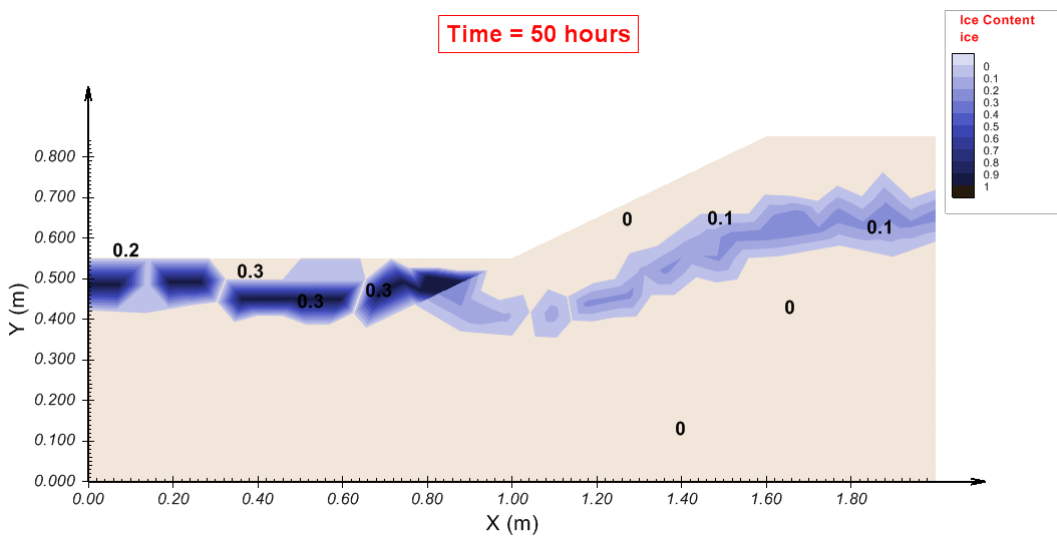
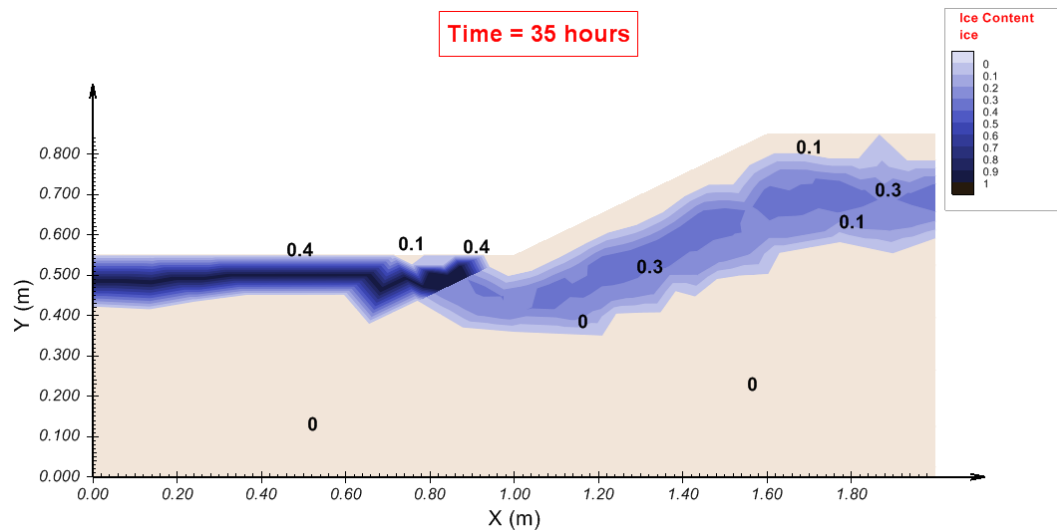




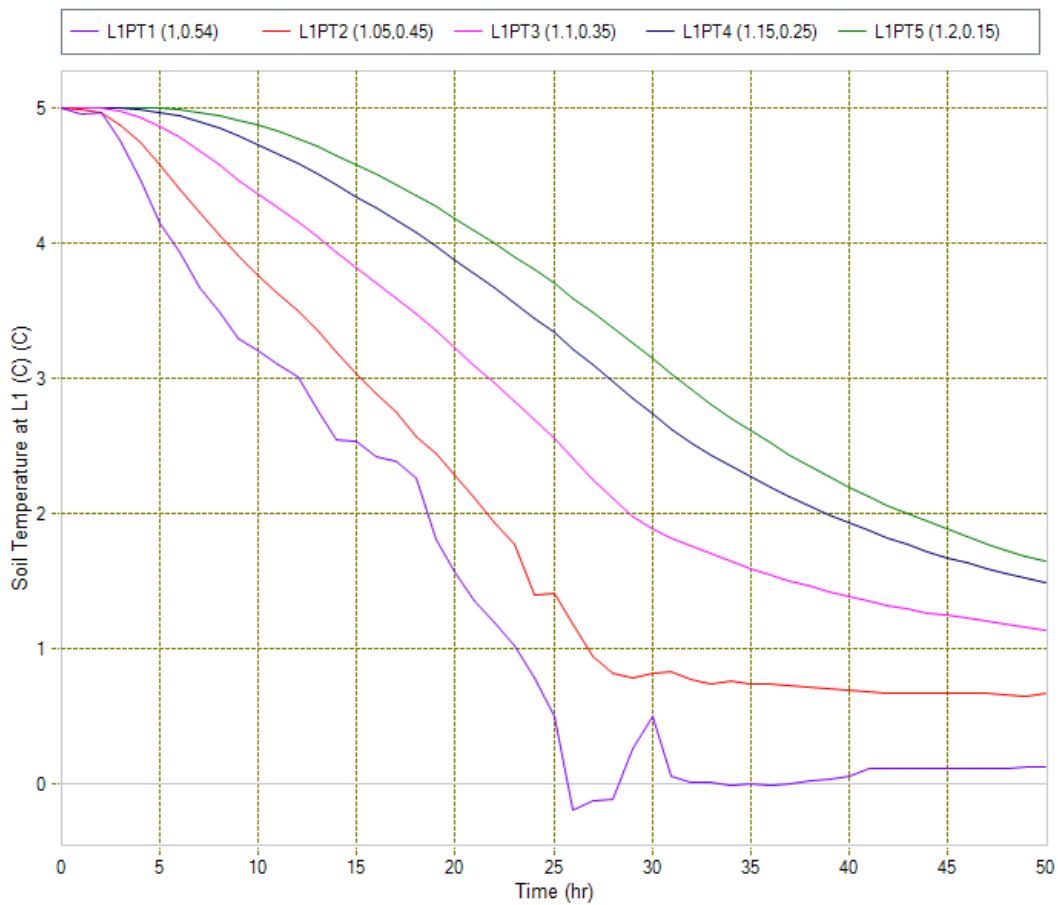


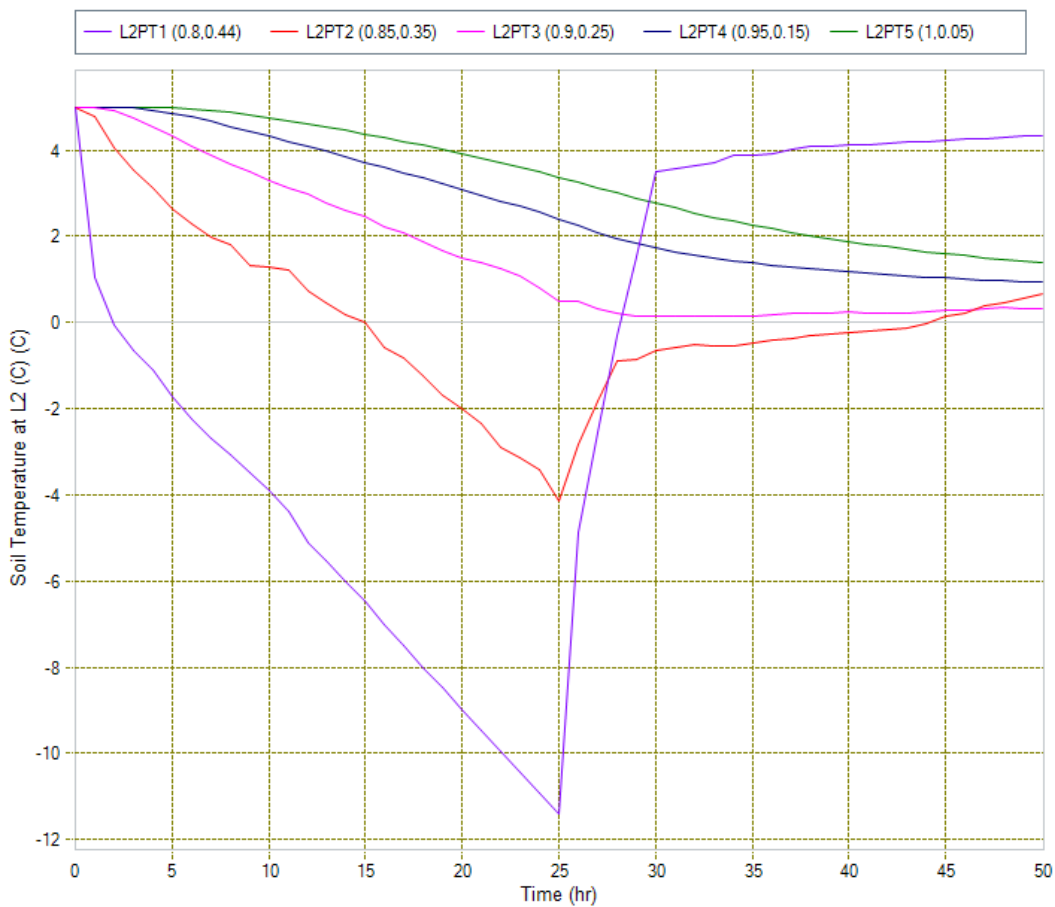
The following screenshots show contours of the fraction of ice content during freezing and thawing of the water and canal bank.

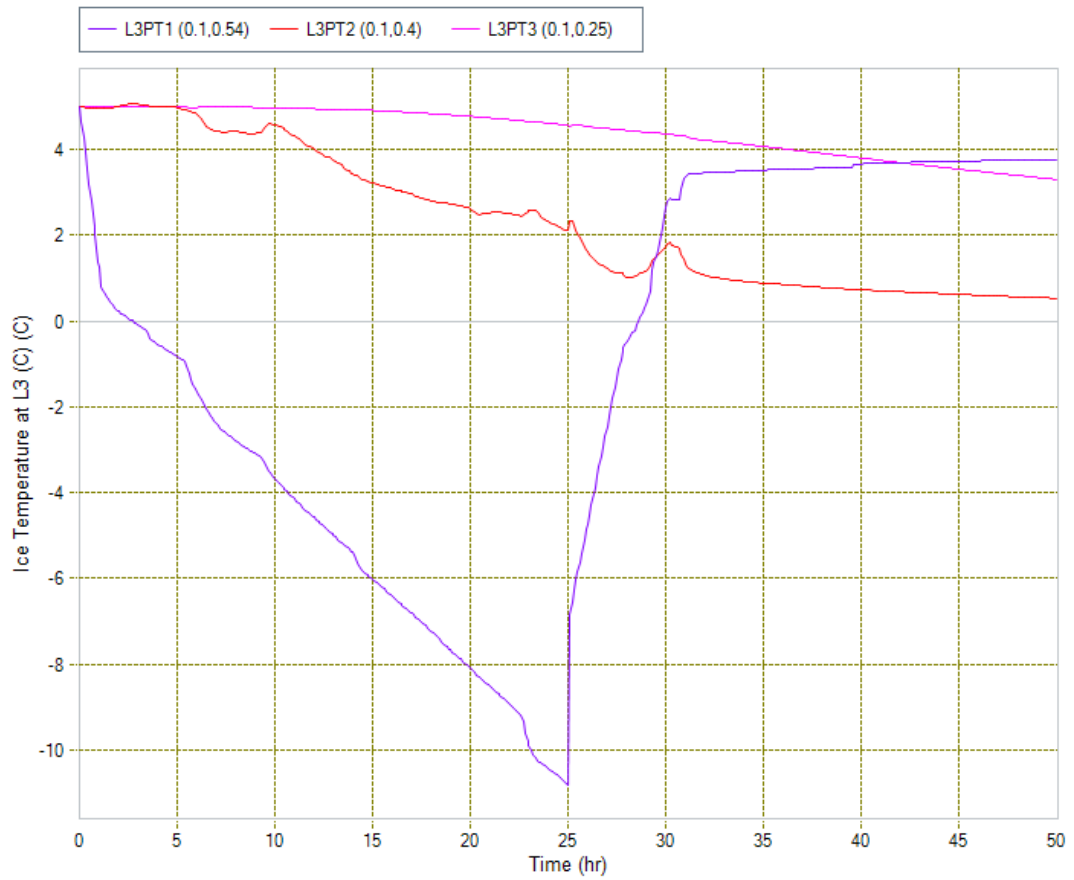




The history plots for the measured temperatures along *L1*, *L2*, and *L3* are shown below.





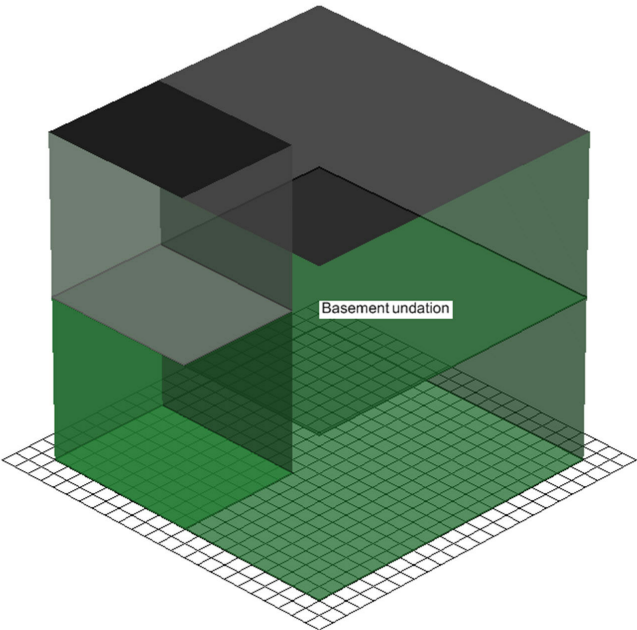


# 10 3D Heated Foundation

The following example will introduce you to a three-dimensional model in SVHEAT. The model is used to investigate steady-state heat flow through a material resulting from a heated foundation under winter conditions. The tutorial provides a detailed set of instructions guiding the user through the creation of the 3D heat transfer model. The model is generated using two regions, three surfaces, and one material. The model data and material properties are provided below.

Project: Foundations  
Model: HeatedFoundTut3D  
Minimum authorization required: STANDARD

## Model Description and Geometry



## Region Shape Data

Ground Region		Basement Region	
X	Y	X	Y
0	0	12	0
12	0	20	0
20	0	20	10
20	10	12	10
20	20		
12	20		
0	20		
0	10		

## Material Properties

Temperature (°C)	Conductivity (J/s-m-°C)
-2	2
-1	1.999
0	1
1	1.001
2	1.002

## 10.1 Model Setup

The following steps will be required in order to set up the model described in the preceding section. The steps fall under the general categories of:

- Create model
- Enter geometry
- Specify boundary conditions
- Apply material properties
- Specify model output
- Run model
- Visualize results

### NOTE:

Any values on the dialogs there are not specifically mentioned in the steps below are assumed to be the default values currently present.

### a. Create Model

The following steps are required to create the model:

- Open the *SVOFFICE Manager* dialog,
- Select the project called "UserTutorial". If the project does not exist, create a new project called "UserTutorial",
- Create a new model called "HeatedFoundation3D" by pressing the *New* button next to the list of models. Use the settings shown below when creating the new model:

Application:	SVHEAT
System:	3D
Type:	Steady-State
Units:	Metric
Time Units:	Seconds (s)
- Select the *World Coordinate System* tab (Before entering any model geometry it is best to format the Axes to ensure that the model will fit into the drawing space),
- Enter the World Coordinates System coordinates shown below into the dialog,



x min = -10      x max = 30  
y min = -10      y max = 30  
z min = 0        z max = 20

6. Click the *OK* button to close the *Create New Model* dialog and to create the new model.

The workspace grid spacing needs to be set to aid in defining region shapes. The geometry data for this model has coordinates of a precision of 1m. In order to effectively draw geometry with this precision using the mouse, the grid spacing must be set to a maximum value of 1.

1. The *View Options* dialog will appear,
2. Enter 1 for both the horizontal and vertical spacing, and
3. Click *OK* to close the dialog.

### b. Enter Geometry (Model > Geometry)

Model geometry is defined as a set of regions. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from either .DXF files or from existing models.

This model will be divided into two regions, which are named Ground and Basement. To add the necessary regions follow these steps:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions* from the menu,
2. Change the first *region name* from "R1" to *Ground*. Highlight the name and type new text,
3. Press the *New* button to add a second region,
4. Change the name of the second region to *Basement*, and
5. Click *OK* to close the dialog.

The shapes that define each region will now be created. Note that when drawing geometry shapes the region that is current in the region selector is the region the geometry will be added to. The Region Selector is at the top of the workspace. Refer to the previous section of this manual for the geometry points for each region.

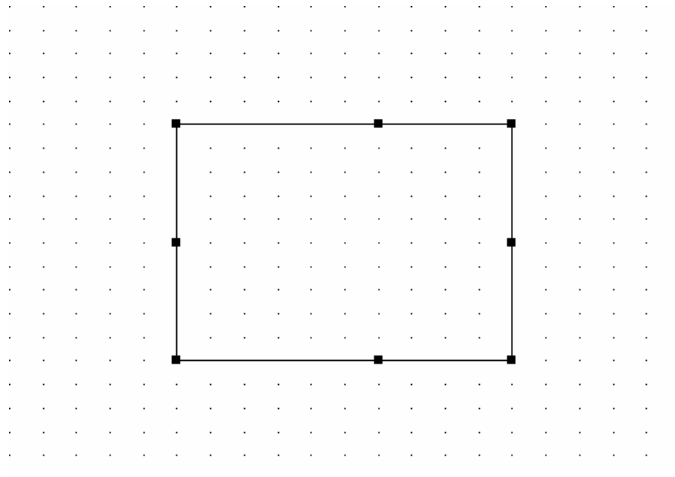
#### • Define the Ground region

1. Select "Ground" as the region in the Region Selector,
2. Select *Draw > Geometry > Polygon Region* from the menu,
3. The cursor will now be changed to cross hairs,
4. Move the cursor near (0,0) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar just below the workspace,
5. To select the point as part of the shape left click on the *Point*,
6. Now move the cursor near (12,0) and then left click on the *Point*. A line is now

drawn from (0,0) to (12,0),

7. Repeat this process for the remaining points. Refer to the previous section of this manual for the geometry points for each region,
8. For the last point (0,10). Double-click on the point to finish the shape. A line is now drawn from (0,20) to (0,10) and the shape is automatically finished by SVHEAT by drawing a line from (0,10) back to the start point, (0,0).

If the ground region geometry been entered correctly the shape should look like the following:



#### NOTE:

If a mistake was made entering the coordinate points for a shape then select a shape with the mouse and select *Edit > Delete* from the menu. This will remove the entire shape from the region. To edit the shape use the *Region Properties* dialog.

#### • Define the Basement

9. Select "Basement" in the Region Selector,
10. Select *Draw > Geometry > Region Polygon* from the menu,
11. Move the cursor near (12,0) in the drawing space. You can view the coordinates of the current position of the mouse in the status bar just below the drawing space,
12. To select the point as part of the shape left click on the *Point*,
13. Now move the cursor near (20,0) and then left click on the *Point*. A line is now drawn from (12,0) to (20,0),
14. Now move the cursor near (20,10) and then left click on the *Point*. A line is now drawn from (20,0) to (20,10), and
15. For the last point (12,10). Double-click on the *point* to finish the shape. A line is now drawn from (20,10) to (12,10) and the shape is automatically finished by

SVHEAT by drawing a line from (12,10) back to the start point, (12,0).

This model consists of three surfaces. By default every model initially has two surfaces.

- **Define Surface 1**

This surface is already present with a default constant elevation of 0m.

- **Define Surface 2**

This surface is already present. Follow these steps to set a constant elevation of 10m.

1. Select "Surface 2" in the Surface Selector,
2. Click *Model > Geometry > Surfaces* in the menu to open the *Surfaces* dialog,
3. Click the Properties button to open the *Surface Properties* dialog
4. Enter 10 in the constant elevation box,
5. Click *OK* to *Close* the dialog,

- **Define Surface 3**

The following steps are required to add the third surface to the model:

6. Click "New" to open the *Insert Surfaces* dialog,
7. Press *OK* to add the surface,
8. Click the Properties button to open the *Surface Properties* dialog for Surface 3,
9. Select Constant as the Surface Option,
10. Enter 20 in the constant box,
11. Click *OK* to close the dialog,
12. Click *OK* to close the Surfaces dialog.

### c. Specify Boundary Conditions (*Model > Boundaries*)

Now that all of the regions and surfaces have been successfully defined, the next step is to specify the boundary conditions. A temperature of  $-3^{\circ}\text{C}$  will be defined at the ground surface region to simulate an outdoor temperature. The base of the model will be set to  $-4^{\circ}\text{C}$  to establish a temperature gradient slightly increasing with depth. The basement will be set to a temperature of  $10^{\circ}\text{C}$ .

The steps for specifying the boundary conditions are as follows:

1. Change to the 2D view by selecting *View > Mode > 2D*,
2. Select the Ground region with the mouse, right-click, and choose Boundary Conditions. The *Boundary Conditions* dialog will open and display the boundary conditions for the current Surface,
3. Select "Surface 3" from the drop-down box located at the top right of the dialog,
4. Under the *Surface Boundary Conditions* tab, select a Temperature Constant

- boundary condition from the surface boundary condition drop-down box,
5. Enter -3 in the Constant field.
  6. Select "Surface 1" from the drop-down box located at the top right of the dialog,
  7. Under the *Surface Boundary Conditions* tab, select a Temperature Constant boundary condition from the surface boundary condition drop-down box,
  8. Enter -4 in the Constant field,
  9. Press *OK* to close the dialog,
  10. Specify Basement as the region by selecting *Model > Geometry > Regions* from the menu and clicking on Basement,
  11. Press *OK* to close the dialog,
  12. Select the Basement region with the mouse, right-click, and choose Boundary Conditions. The *Boundary Conditions* dialog will open to the *Sidewall Boundary Conditions* tab and display the boundary conditions for the current Surface,
  13. Select "Surface 2" from the drop-down box,
  14. Select the point (20,10) from the list,
  15. From the Boundary Condition drop-down select a "Temperature Expression" boundary condition from the surface boundary condition drop-down box. This will cause the Expression box to be enabled,
  16. In the Expression box enter a temperature expression of "if  $z > 19$  then -3 + (20-z)\*13 else 10",
  17. Enter BasementWalls as the Boundary Name,
  18. Select the point (12,10) from the list,
  19. From the Boundary Condition drop-down select a "Continue" boundary condition,
  20. Under the *Surface Boundary Conditions* tab, select a Temperature Constant boundary condition from the surface boundary condition drop-down box,
  21. Enter 10 in the Constant field,
  22. Select "Surface 1" from the drop-down box located at the top right of the dialog,
  23. Under the *Surface Boundary Conditions* tab, select a Temperature Constant boundary condition from the surface boundary condition drop-down box,
  24. Enter -4 in the Constant field, and

**NOTE:**

Enter the expression without the quotes shown in the above statement. The purpose of this expression, as opposed to just entering 10, is to smooth the temperature from the ground surface temperature of -3°C to the desired basement temperature of 10°C. This technique removes a potentially sharp break in adjacent boundary conditions and allow the model the converge on a solution much faster.

25. Press *OK* to close the dialog.

#### **d. Apply Material Properties** (Model > Materials)

The next step in defining the model is to enter the material properties for a clay material in the model. This section provides instructions on creating the clay material.

1. Open the *Materials* dialog by selecting *Model > Materials > Manager* from the menu,
2. Click the *New...* button to create a material,
3. Enter "3D Tutorial Soil" for the material name,
4. Press *OK*. The dialog for the new material properties will pop up,

**NOTE:**

When a new material is created, you can specify the display a color for the material using the Fill Color box on the Material Properties dialog. Any region that has a material assigned to it will display the material fill color.

5. Select Data as the Thermal Conductivity Option On the *Conductivity* tab,
6. Click on the *Data* button,
7. Enter the *data points* as provided in the opening section at the beginning of this tutorial,

**NOTE:**

To view the data press the *Graph* button.

8. Click the *Volumetric Heat Capacity* tab,
9. Select the *Constant option*,
10. Check the Same Value of Unfrozen and Frozen HC option
11. Enter 1.95e6 for the Frozen VHC text field, and
12. Click *OK* on all opened dialogs to accept the changes and close the dialogs.

Each region will cut through all the layers in a model creating a separate "block" on each layer. Each block can be assigned a material or be left as *void*. A void area is essentially air space.

1. Select *Model > Materials > Material Layers* from the menu to open the *Material Layers* dialog,
2. Use the arrow buttons to select the Ground region,
3. Select the "3D Tutorial Soil" material from the drop-down for Layer 2,
4. Select the "3D Tutorial Soil" material from the drop-down for Layer 1,
5. Select "Basement" region by using the right arrow in the top right of the dialog,
6. Select "VOID" from the drop-down for *Layer 2*,
7. Select the "3D Tutorial Soil" material from the drop-down for *Layer 1*, and
8. Close the dialog using the *OK* button.

## e. Specify Model Output

### PLOT MANAGER (Model > Reporting > Plot Manager)

The *Plot Manager* dialog, when first opened will display the default plots. There are many plot types that can be specified to visualize the results of the model. A temperature surface plot will be added in addition to the default plots.

1. Open the *Plot Manager* dialog by selecting *Model > Reporting > Plot Manager* from the menu,
2. The Plot Manager dialog will open with the default plots present,
3. The toolbar at the bottom left corner of the dialog contains a button for each

- plot type. Click on the *Surface* button. The *Plot Properties* dialog will open,
4. Enter the title *TempSurface*,
  5. Select "Te" as the variable to plot from the drop-down,
  6. Move to the *Projection* tab,
  7. Select "Plane" as the Projection Option,
  8. Select "Z" from the Coordinate Direction drop-down,
  9. Enter 10 in the Coordinate field. This will generate a 2D slice at  $Z = 10\text{m}$  on which the temperature will be plotted,
  10. Click *OK* to close the dialog and add the plot to the list,
  11. The default plots were generated to have a projection (x or y coordinate) at the midpoint of the model. The basement wall is at  $X = 12\text{m}$  so the plots with a  $X=10\text{m}$  coordinate will be adjusted to  $X=12\text{m}$ ,
  12. Select the plot with the title *YZ Temp* and click the Properties button,
  13. Select the Projection tab and adjust the Coordinate from 10 to 12,
  14. Click *OK* to close the dialog,
  15. Repeat the previous 3 steps for all the plots with an X coordinate of 10m, then
  16. Click *OK* to close the *Plot Manager* and return to the workspace.

#### OUTPUT FILES (Model > Reporting > Output Manager)

Two output files will be generated for this tutorial model: a transfer file of temperatures, and a .dat file to transfer the results to ACUMESH. The default *AcuMeshInput.dat* file is listed in the Output Manager by default for every model. It contains the most common variables used for visualization in *AcuMesh*.

1. Open the *Output Manager* dialog by selecting *Model > Reporting > Output Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each output file type. Create the Te transfer file by pressing the *SVHeat* button, and
3. Click *OK* to close the *Output Manager* and return to the workspace.

#### f. Run Model (Solve > Analyze)

The next step is to analyze the model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the *SVHEAT* solver. The solver will automatically begin solving the model.

#### g. Visualize Results (Window > AcuMesh)

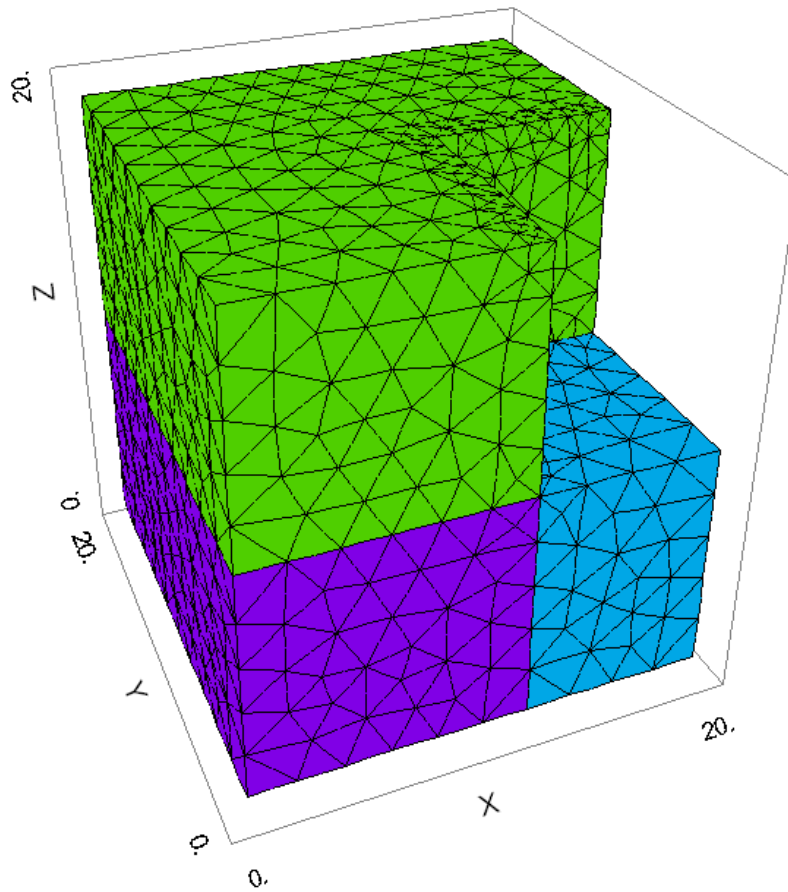
Once you have analyzed the model, the output plots can be visualized using *ACUMESH*. In order to view plots in *ACUMESH*, select *Window > ACUMESH* from the menu. Plots can be visualized by selecting the desired process under Plots in the menu.

## 10.2 Results and Discussion

After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the *SVHEAT* solver.

Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots. Shown

below is the solution mesh generated by the solver.





## 11 References

FlexPDE 6.x Reference Manual, 2007. PDE Solutions Inc. Spokane Valley, WA 99206.

---

This page has been left intentionally.